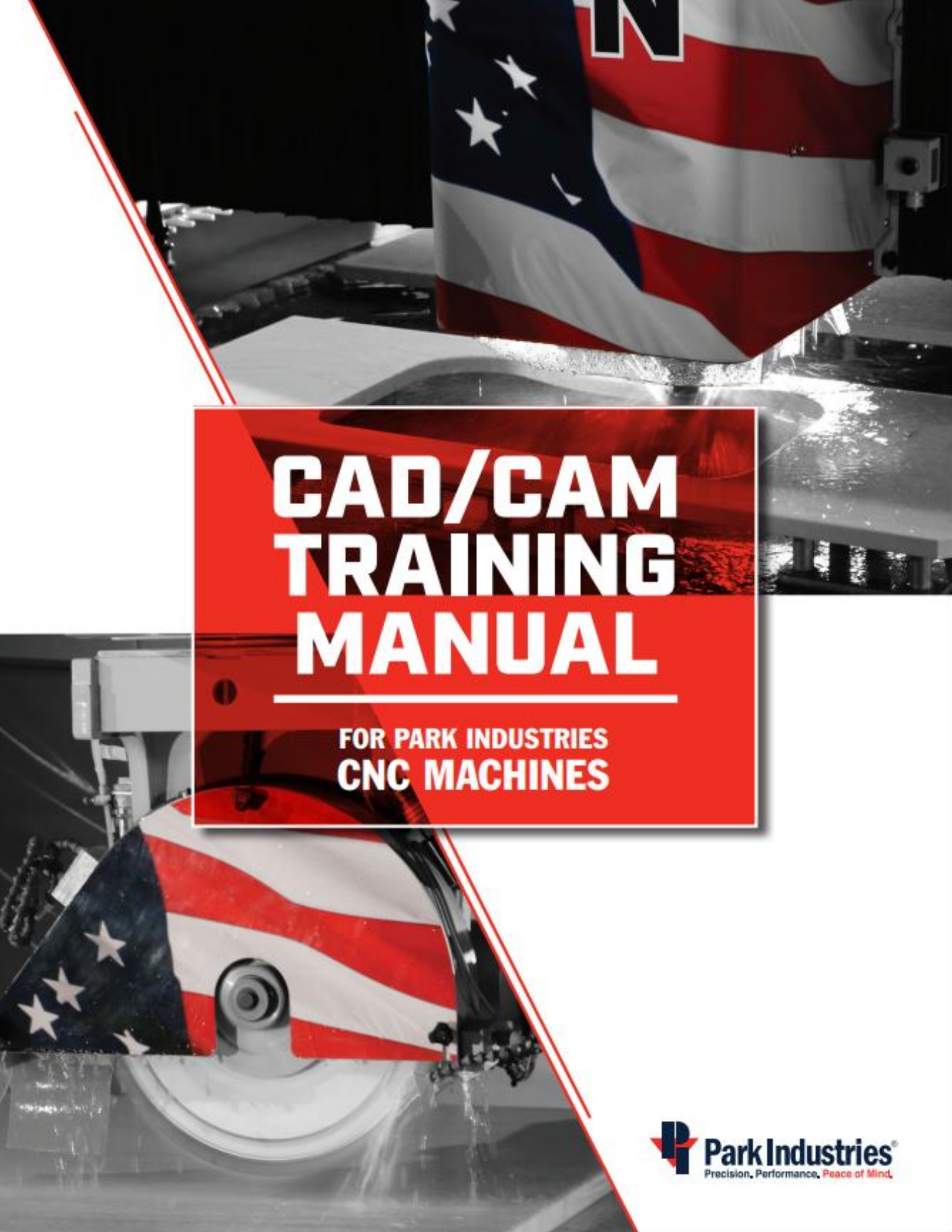


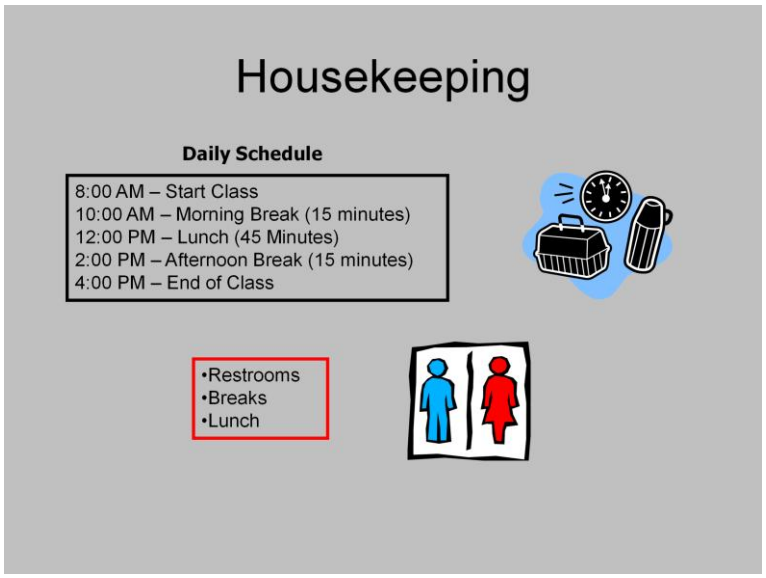
CAD/CAM TRAINING MANUAL

FOR PARK INDUSTRIES CNC MACHINES

Park Industries®
Precision. Performance. Peace of Mind.

Park Industries®

Precision. Performance. Peace of Mind.®



Classroom Computer Password _____

Version 4.18.2025

Table of Contents

Section 1. Alphacam Commands & CAD Examples

(Pages 10- 38)

Section 2. Sink and Digital Template Cleanup

(Pages 39 - 45)

Section 3. Tool Drawing Information

(Pages 46 - 58)

Section 4. Building the Tool Library for Titan

(Pages 59-67)

Section 5. Creating Tool Paths with “Cut Shape"

(Pages 68-78)

Section 6. Titan (Using, Creating and Editing Styles)

(Pages 79 - 86)

Section 7. Fab Center Saw

(Pages 87 - 91)

Section 8 Voyager Saw

(Pages 92 - 96)

Section 9 SABERjet and Javelin Saws

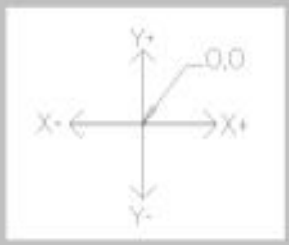
(Pages 97 - 101)

Section 10 Note Pages & Pull-outs

(Pages 102 - 109)

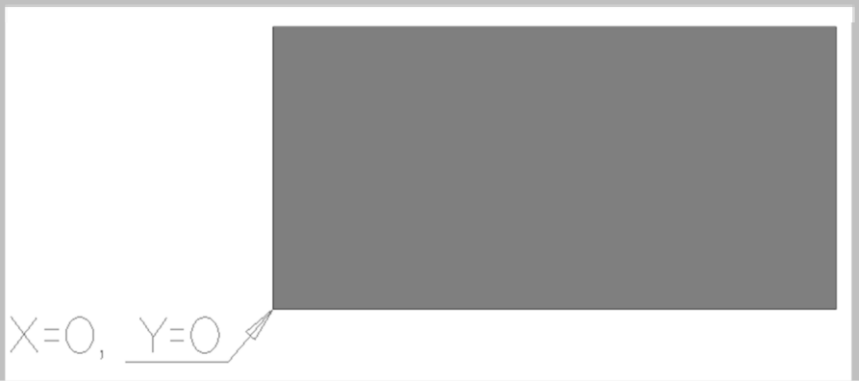
Terminology

- CAD – Computer Aided Design
- CAM – Computer Aided Manufacturing
- CNC – Computer Numerical Control
- Origin – X=0, Y=0, Z=0 In CAD, also known as zero – zero, or zero point



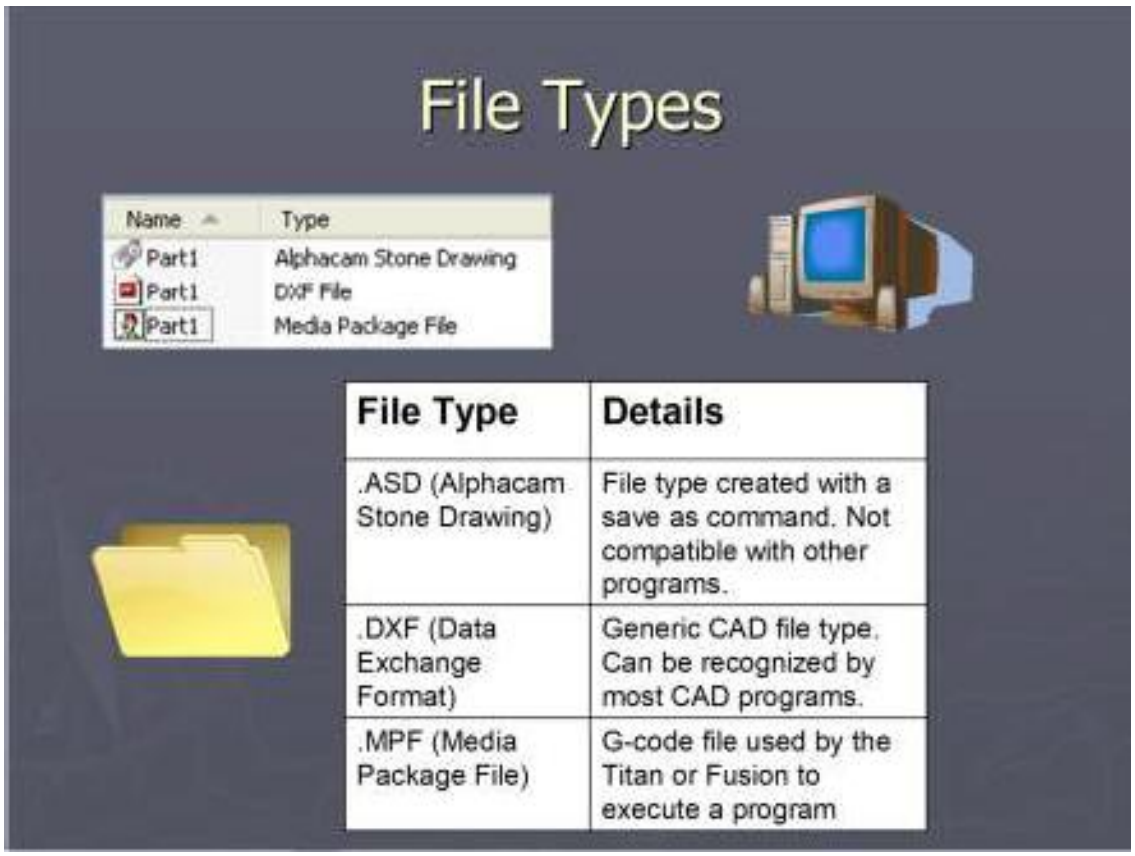
Machine X,Y

- When running a program, the X=0, Y=0 on the machine is the front left corner of the table
- Any moves towards the center of the table are a positive
- This point matches the origin in Alphacam



Files

There are three basic file types that are used in Alphacam for drawing and programming your Park Industries machine.



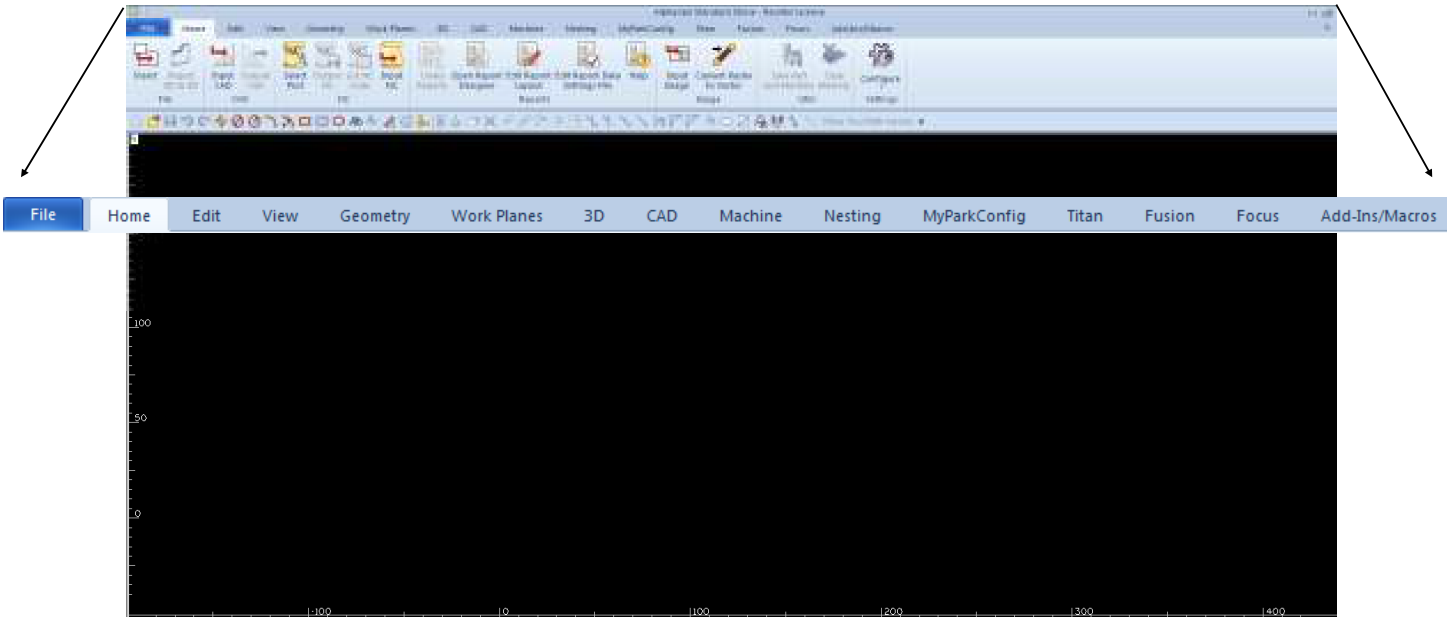
ASD file type is being used when choosing: **NEW, OPEN, SAVE,** or **SAVE AS** from the **FILE** tab, and **INSERT** from the **HOME** tab.

DXF file type is used when choosing: **INPUT CAD** or **OUTPUT CAD** from the **HOME** tab.

MPF file type is created when choosing: **Send G-CODE** from the correct **Park Industries Machine** tab (TITAN, SABERjet, VOYAGER, etc.) There is also a **DXF** file that is sent to the machine to display the program image on the control screen.

Alphacam Ribbon Bar

The “Tabs” at the top of the screen are used to change between command types. Choose the Tab that relates to the function you would like to perform, then select the desired command to complete the task.



Park Industries Toolbar

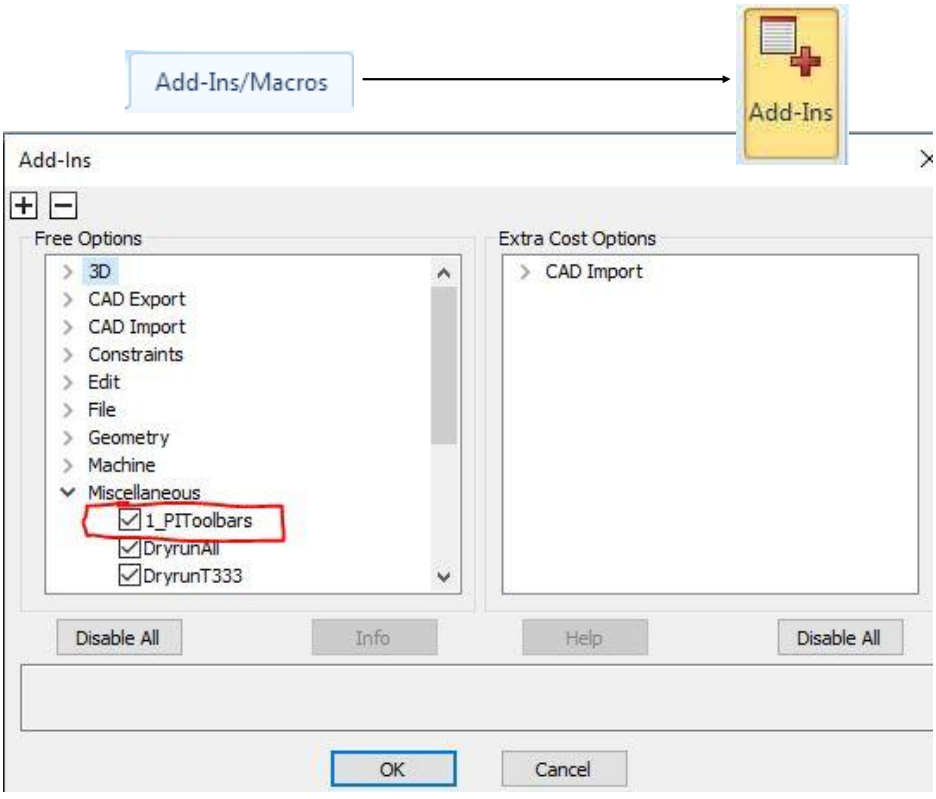


The Quick Access Toolbar with common commands will be installed by Park Industries as an Add-In.

Add-Ins

The Park Machine tabs (Titan, Voyager, etc….) are added to Alphacam by Park Industries. If the machine tabs are not present, choose the **Add-Ins/Macros** tab from the Ribbon bar, and then the **Add-Ins** command from the ribbon.

Place a check in the box next to **1_PIToolbars** then click OK. The **1_PIToolbars** will load the Park Industries machine tabs.



Notes:

Section 1

Alphacam Commands & CAD examples

Videos for this section can be found on our website's service and training guides.
https://www.parkindustries.com/stone/service-support/guides/

Suggested videos/guides include

INTRODUCTION TO CAD

ALPHACAM BASIC CAD DRAWING

DRAWING LINES AT ANGLES (VIDEO)

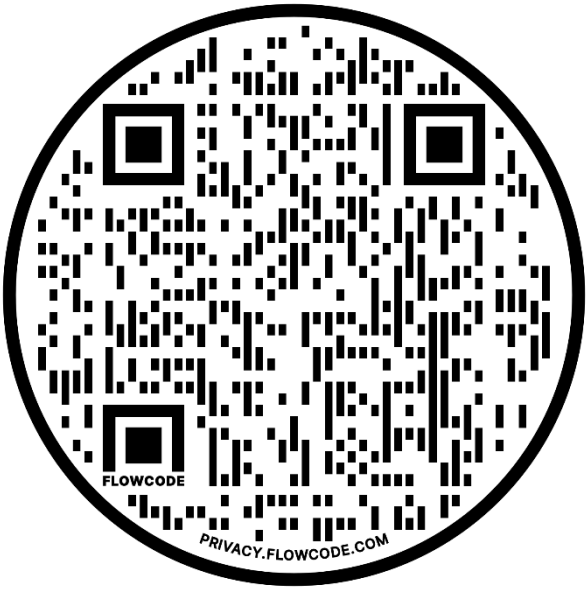
DRAWING TANGENT CIRCLES AND ARCS (VIDEO)

Other suggested videos for Alphacam

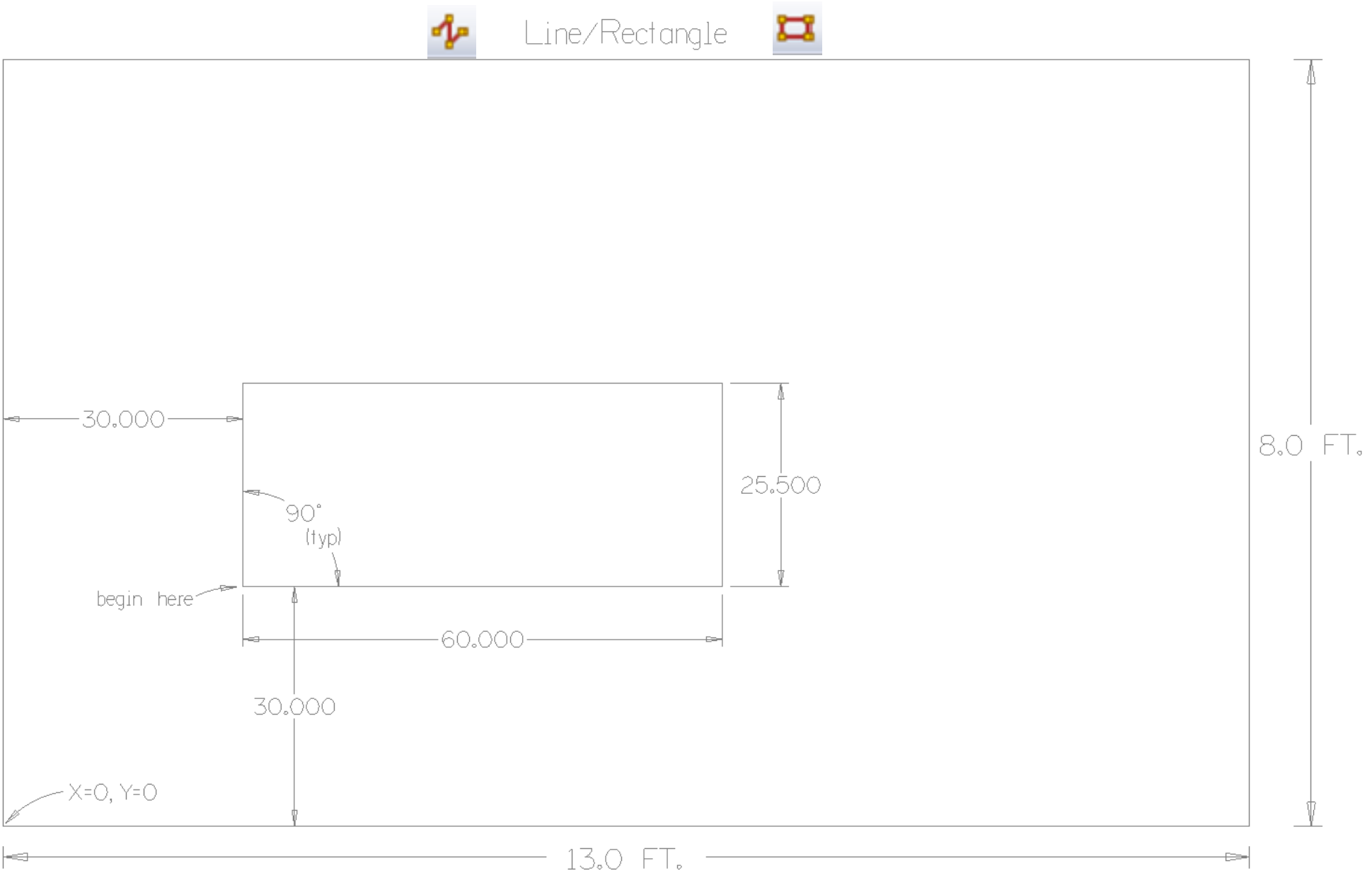
USING AUTOSAVE (VIDEO)

RESTORING RIBBON TABS & KEYBOARD SHORTCUTS (VIDEO)

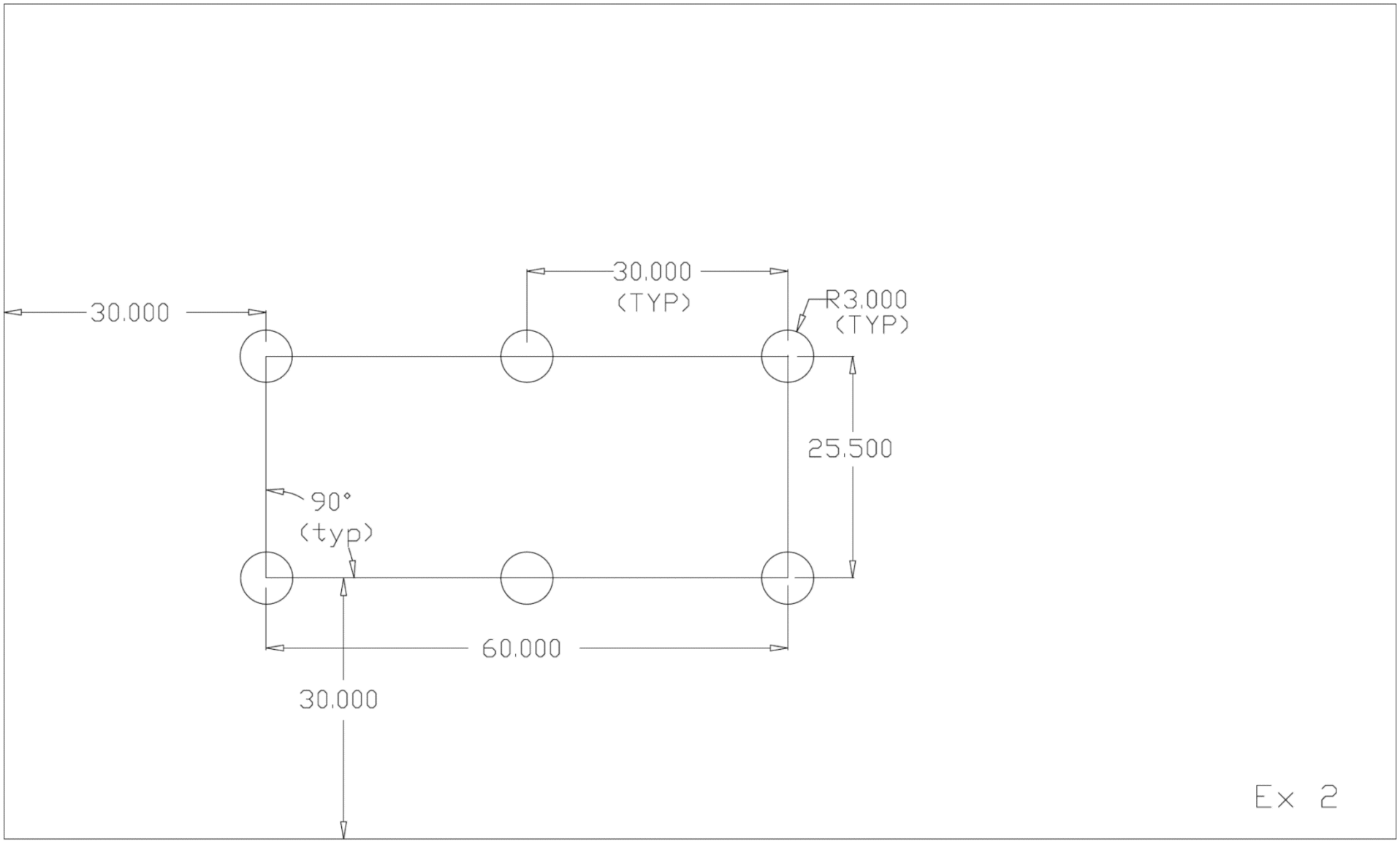
CUSTOMIZING THE TOOLBAR IN ALPHACAM (VIDEO)





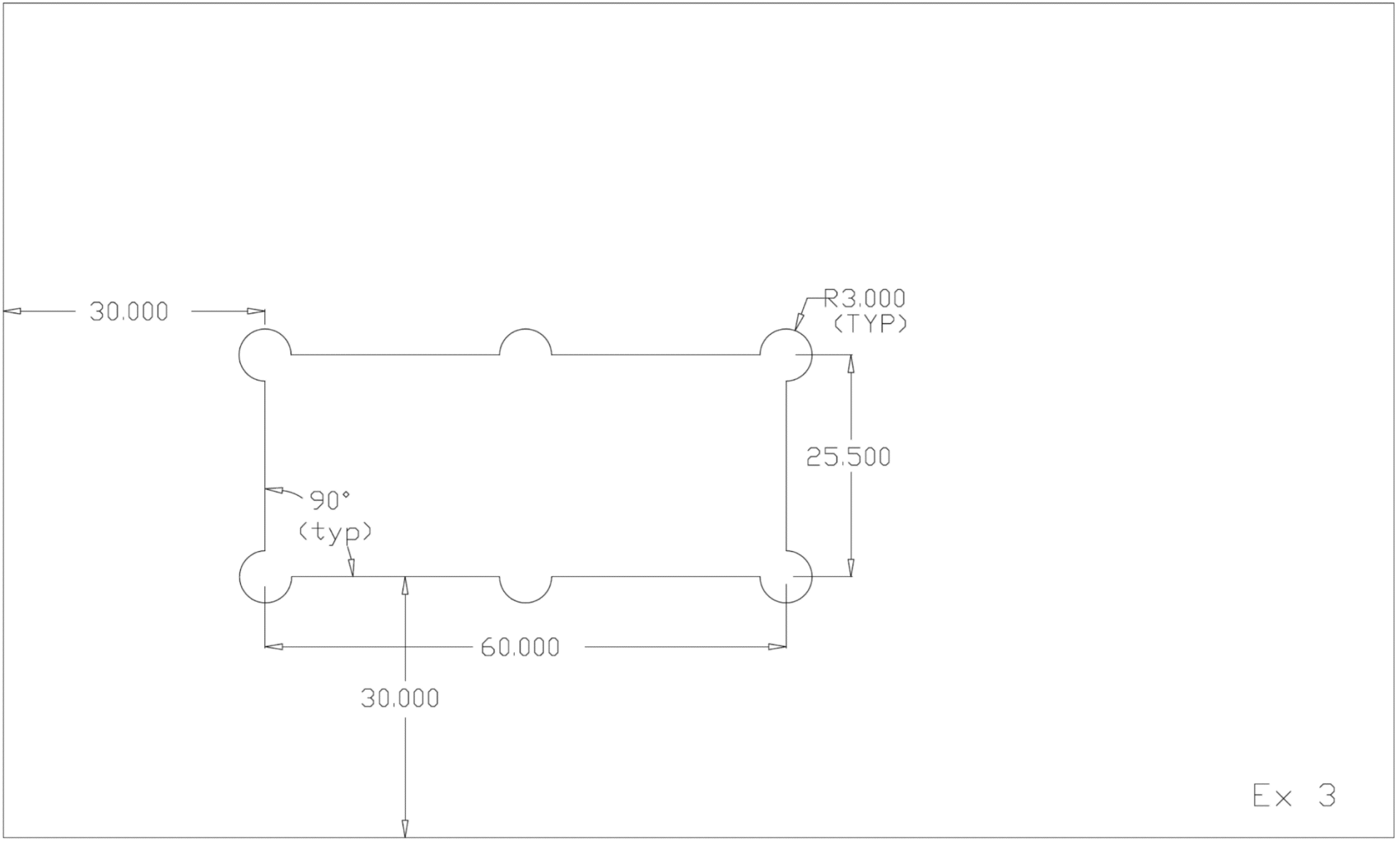

Line/Rectangle
30.000
25.500
90°
(typ)
begin here
60.000
30.000
X=0, Y=0
8.0 FT.
13.0 FT.

Circle+Rad/Dia

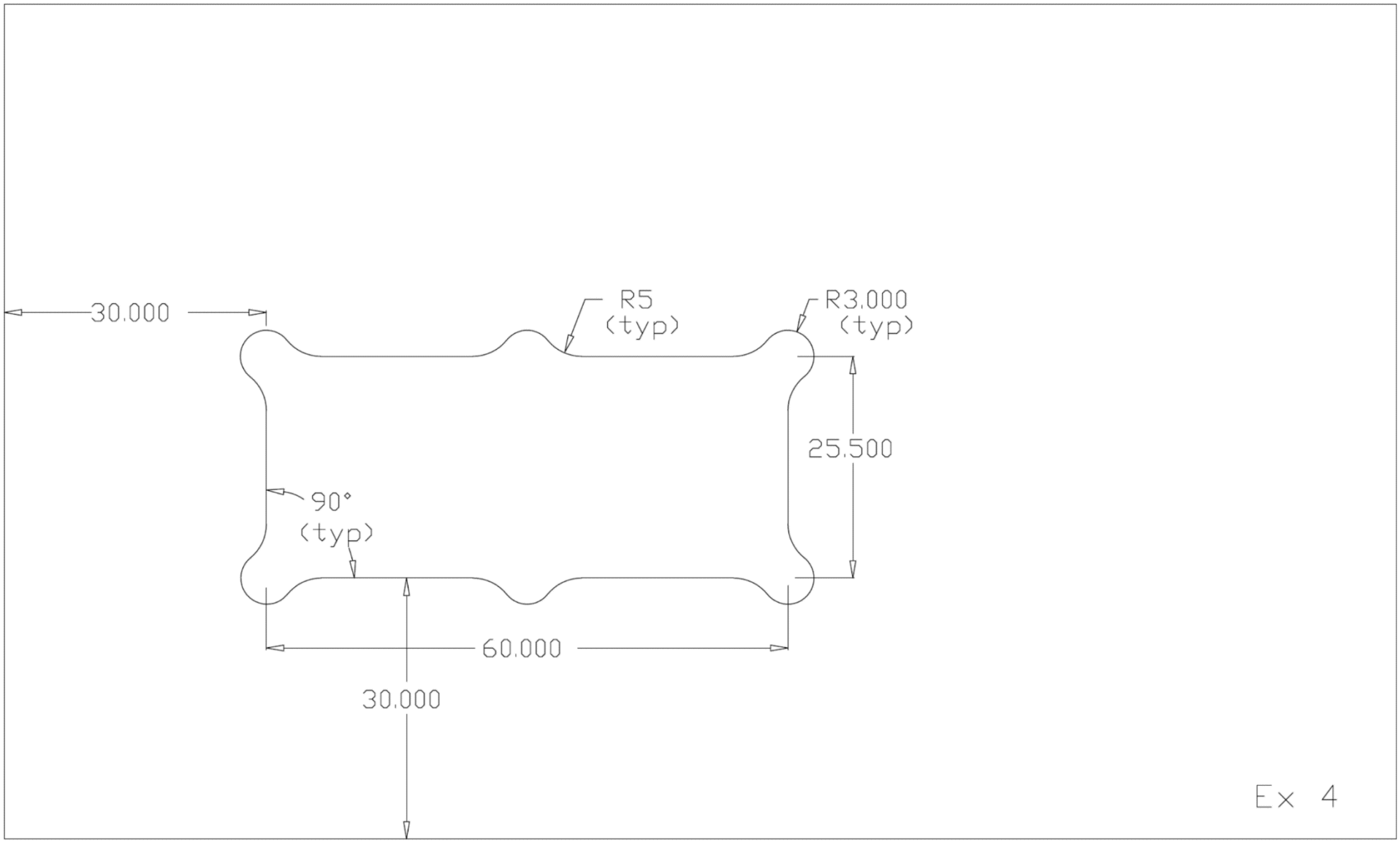




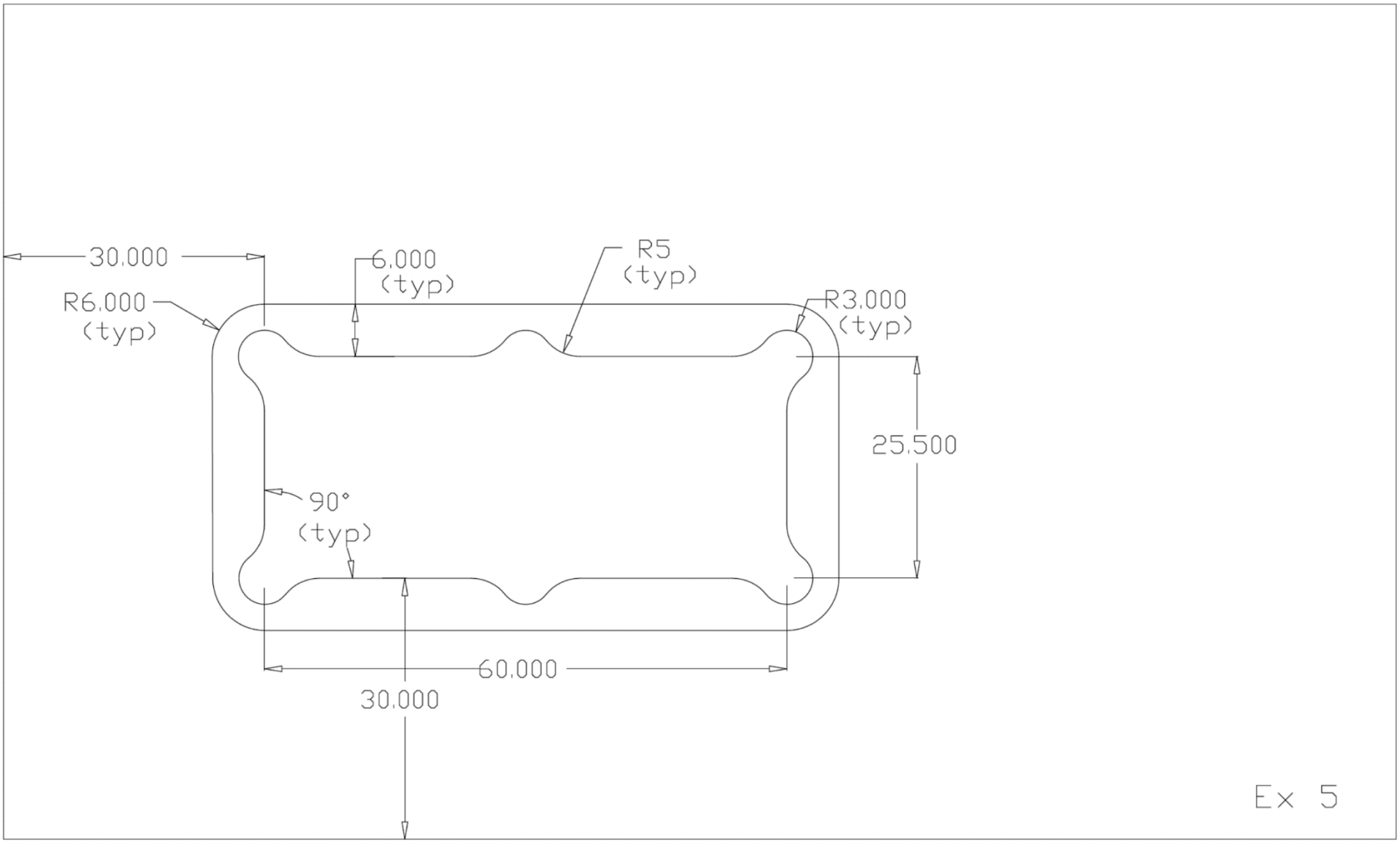
Trim



Fillet

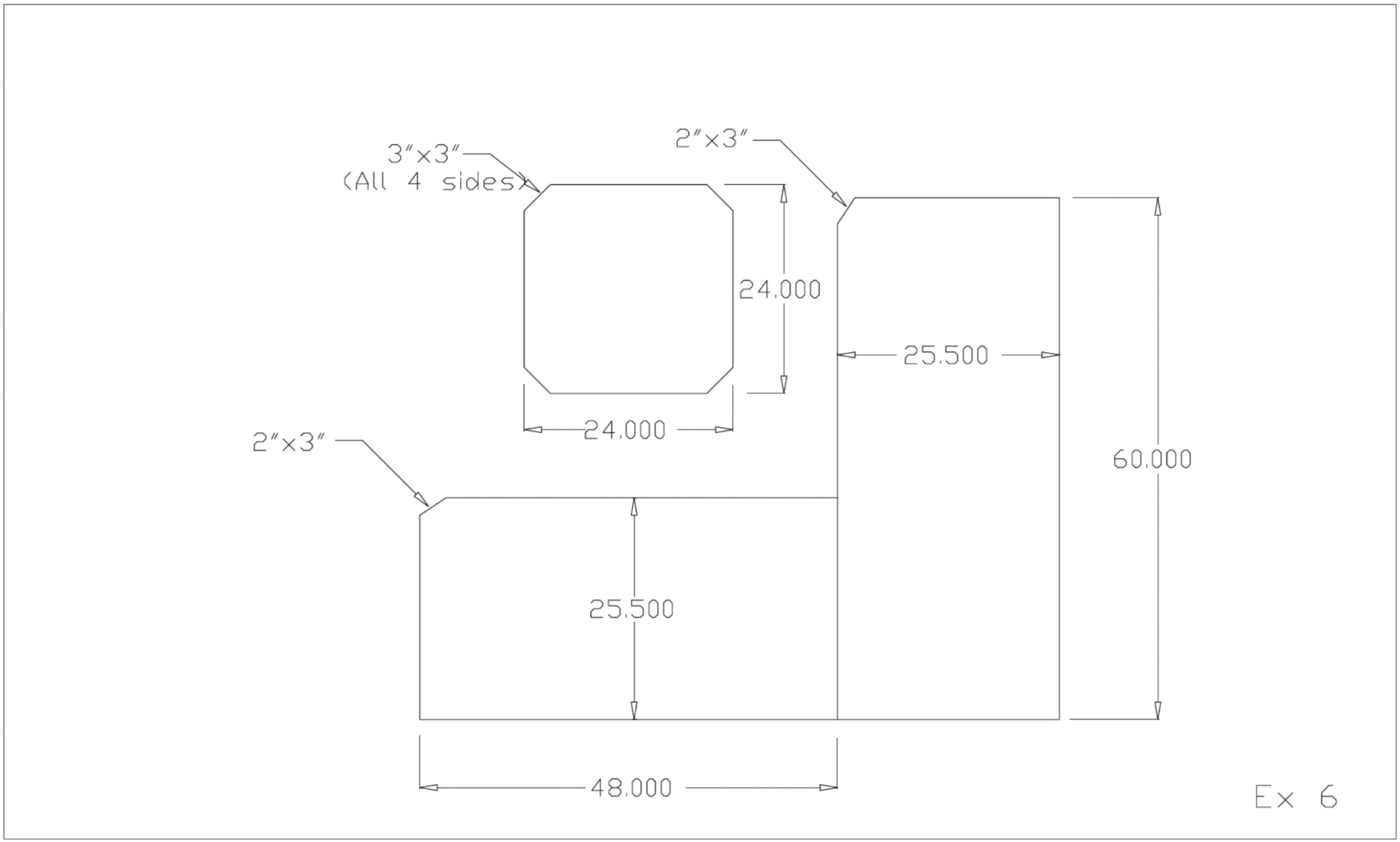


Offset-Fillet

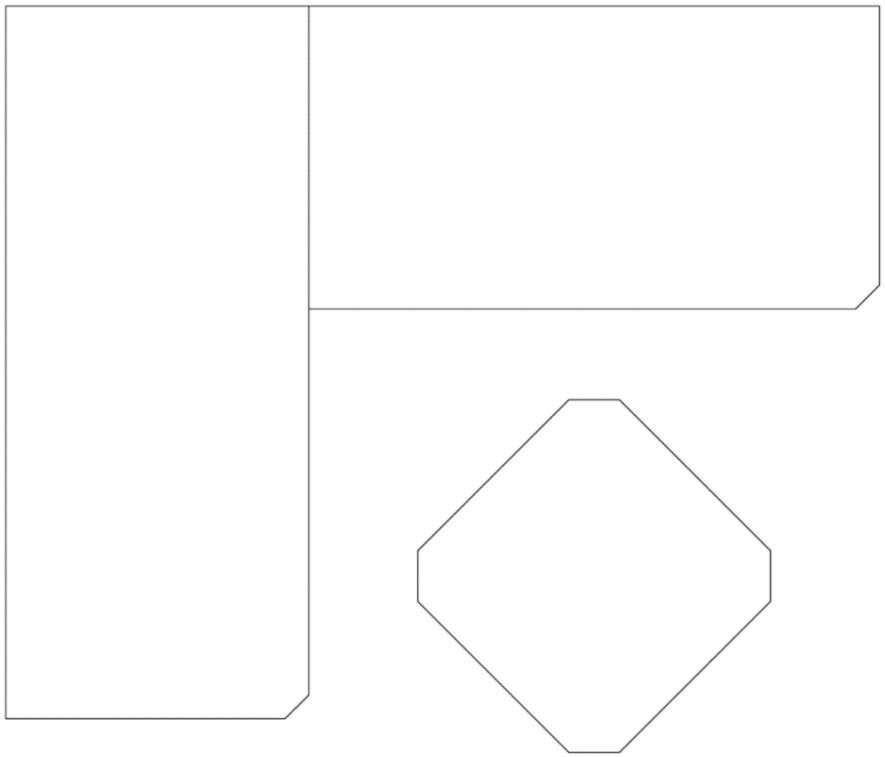




Chamfer

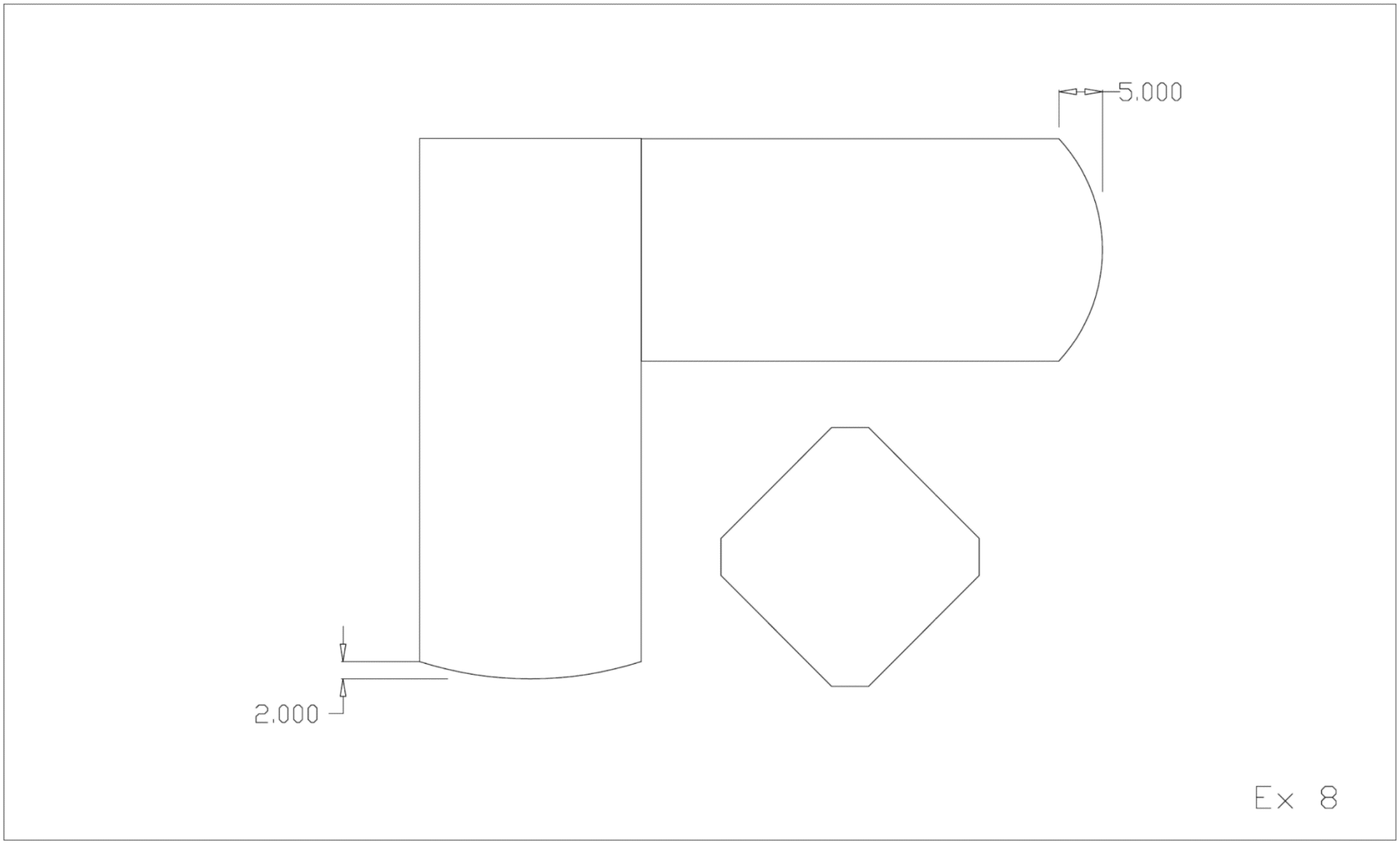

3"x3"
(All 4 sides)
2"x3"
24.000
25.500
24.000
2"x3"
60.000
25.500
48.000
Ex 6

Rotate

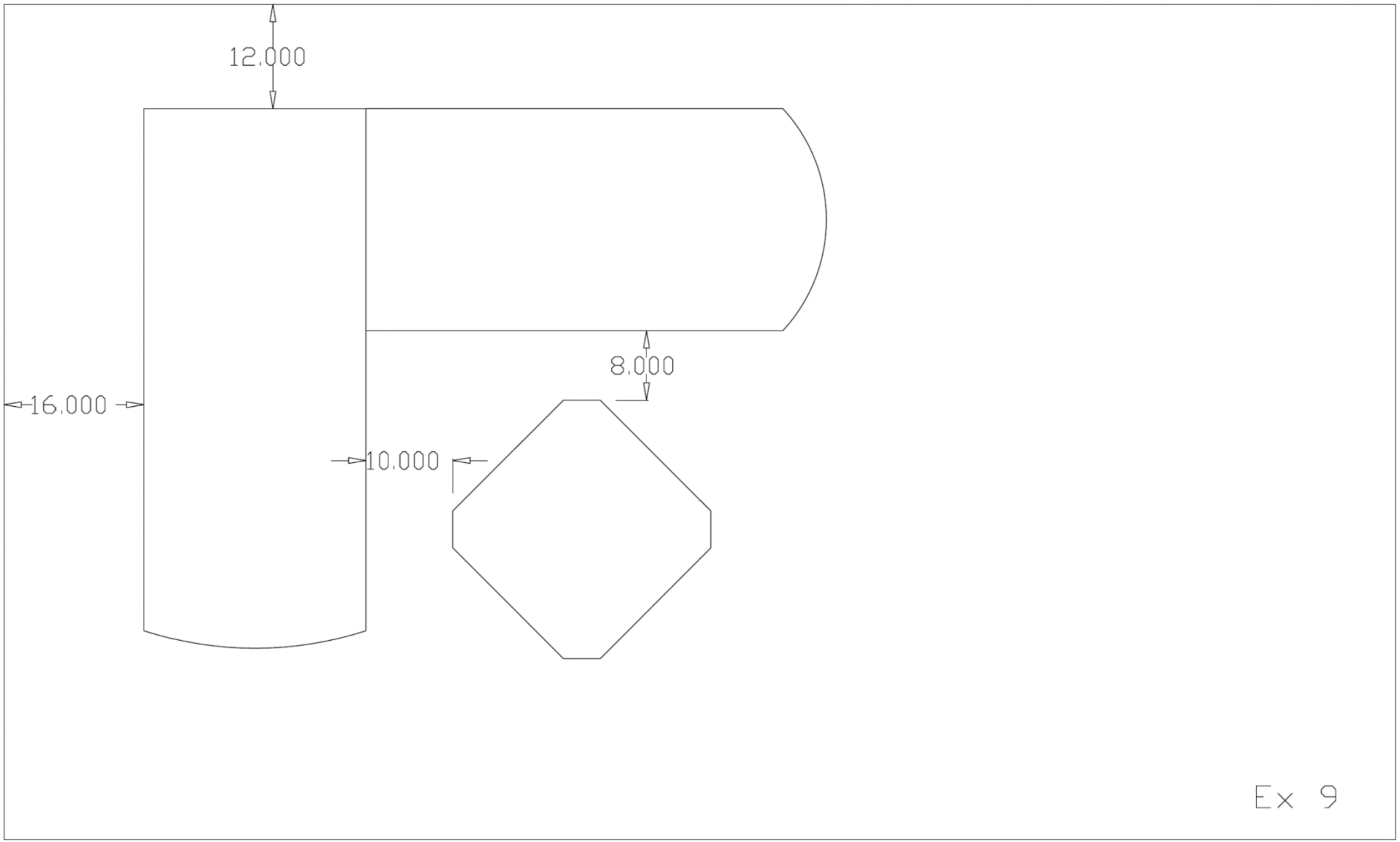


Ex 7

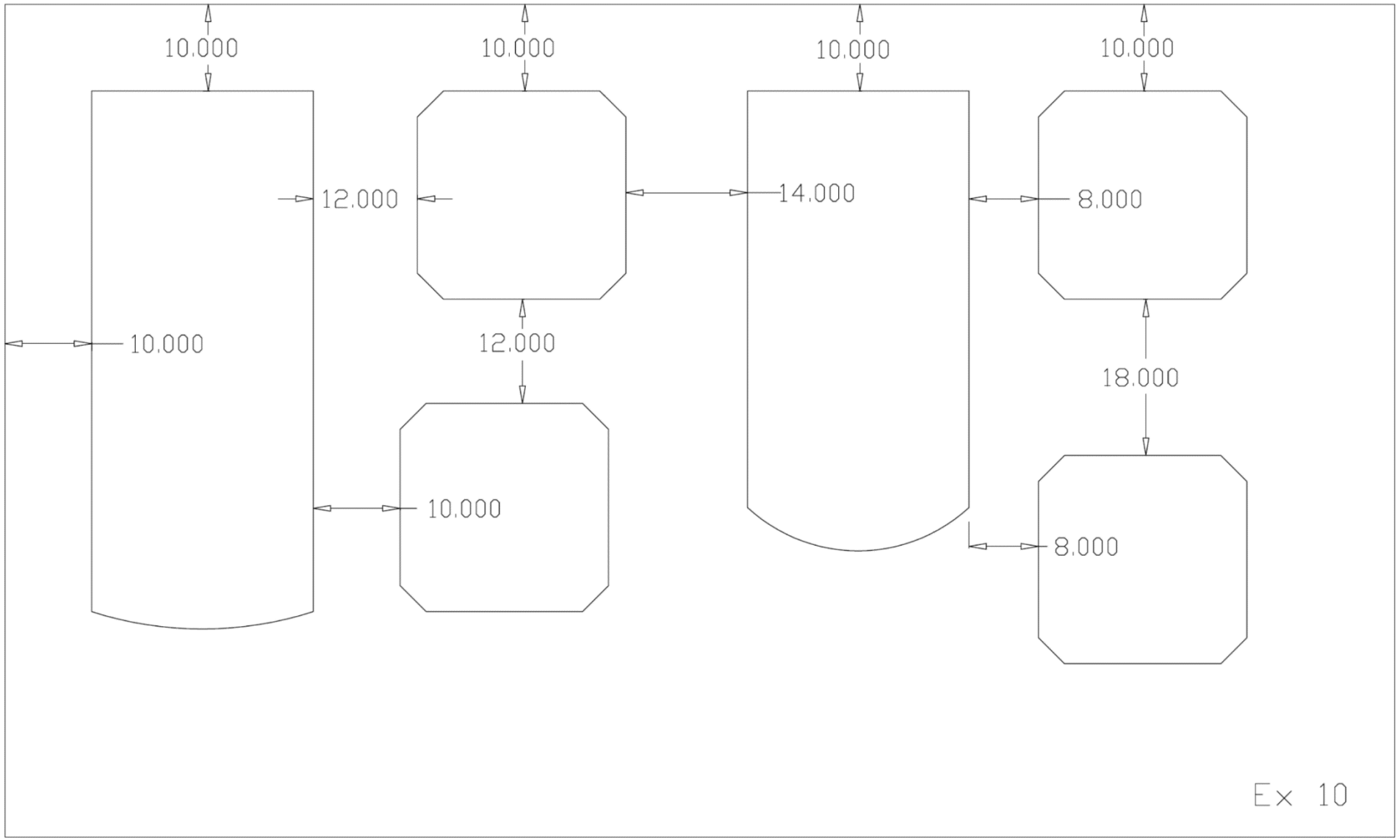
Offset+Arc



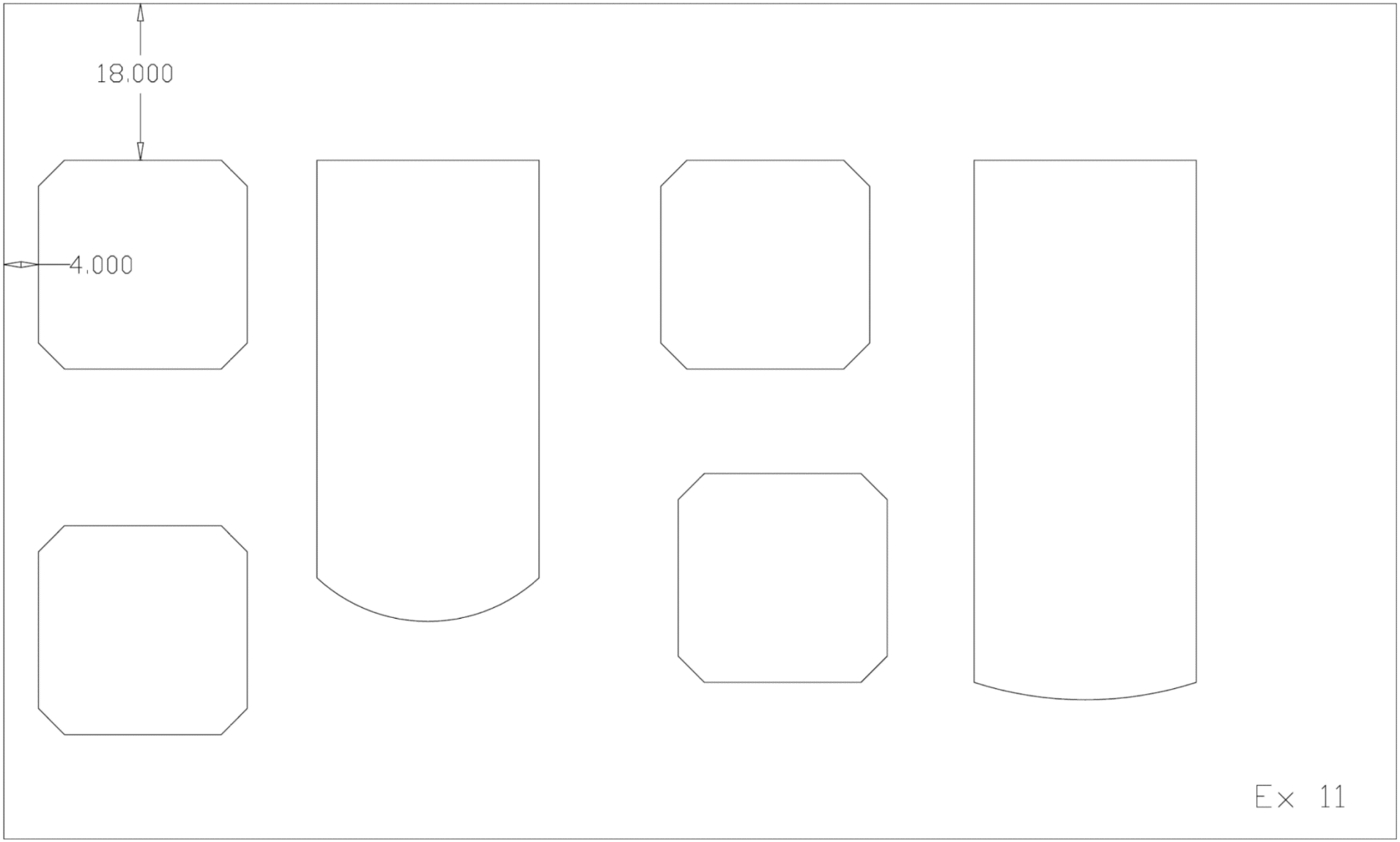
Offset+Move



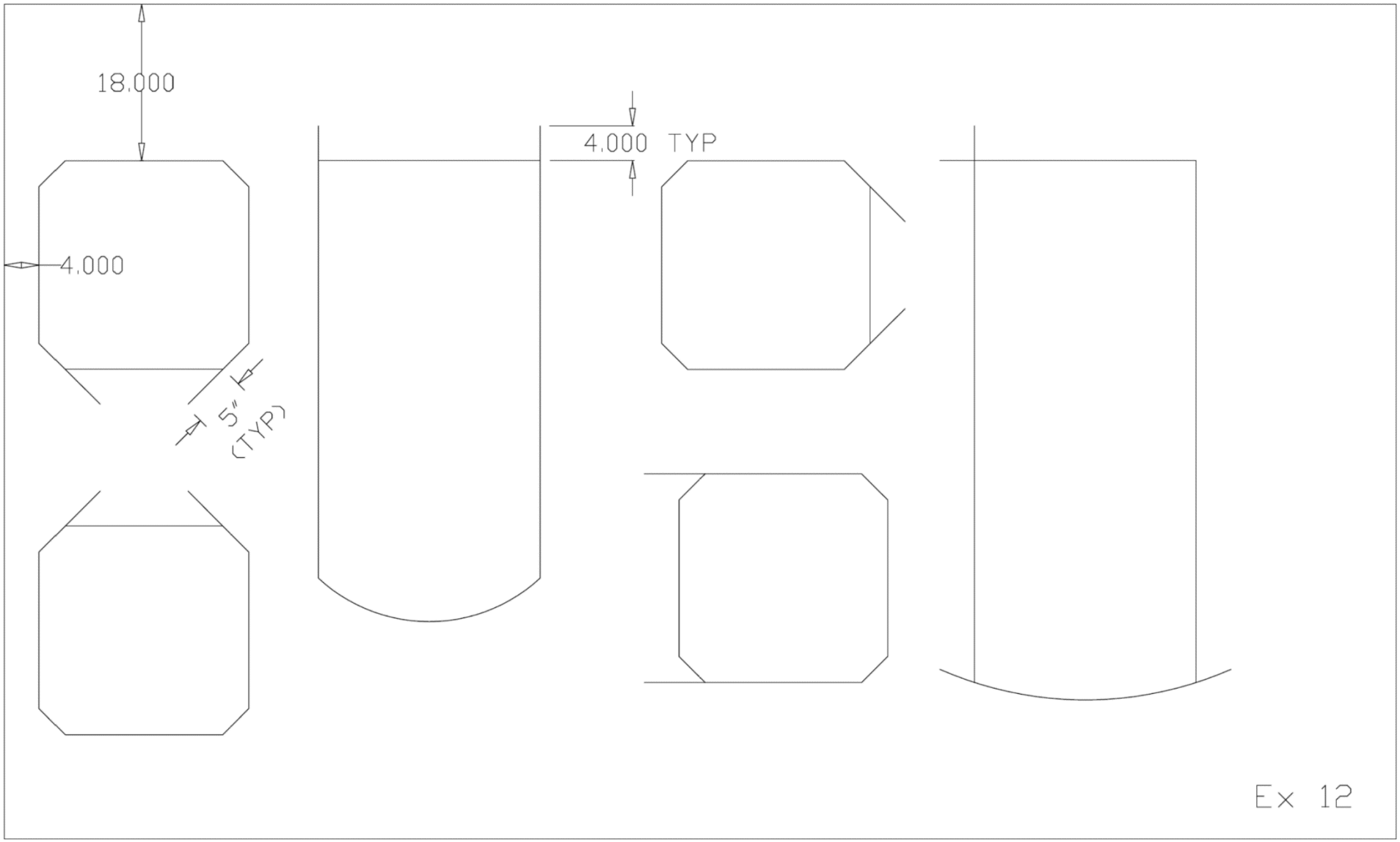
Offset+Copy+Move

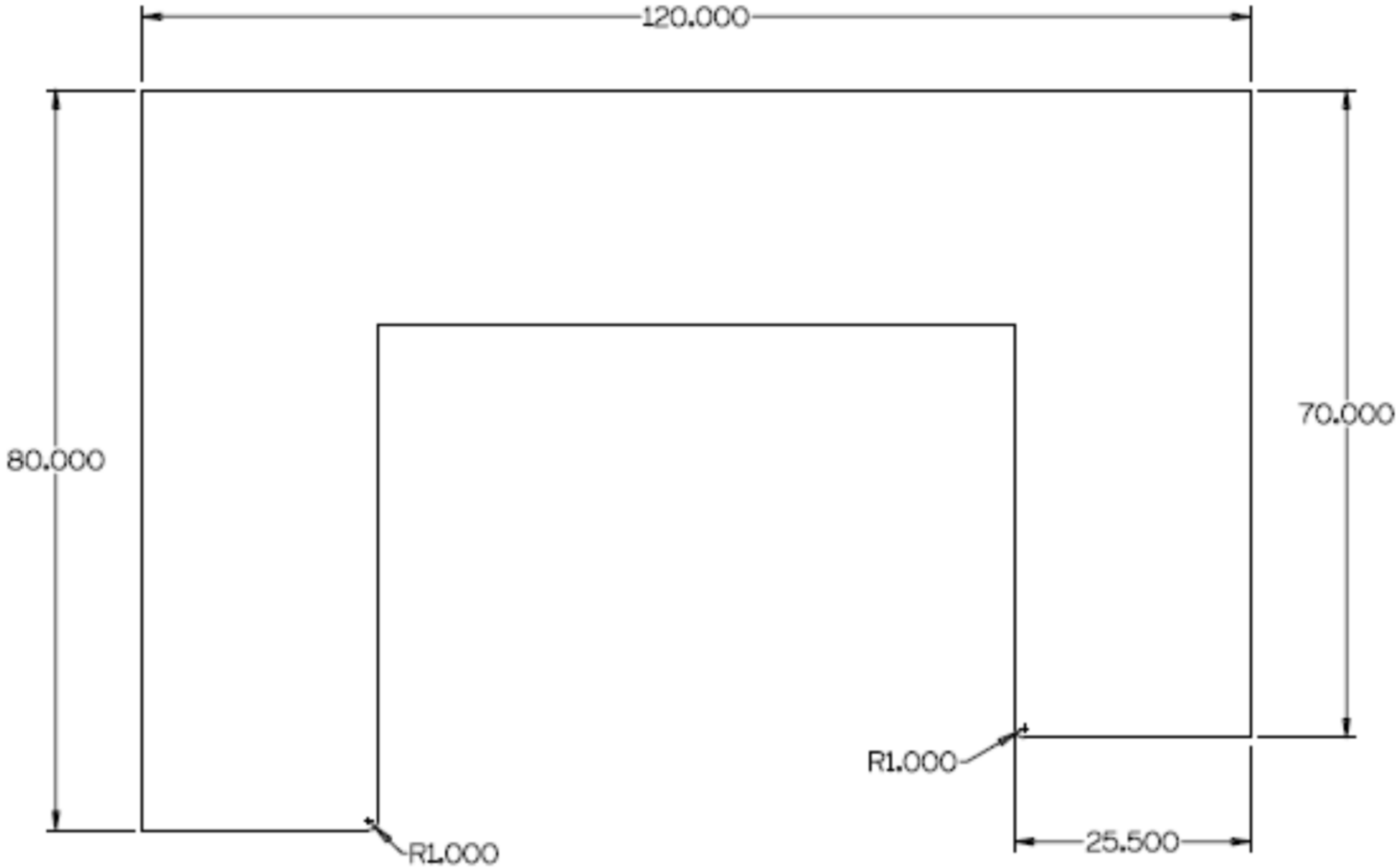


Mirror + Move



Offset + Extend/Extend by distance



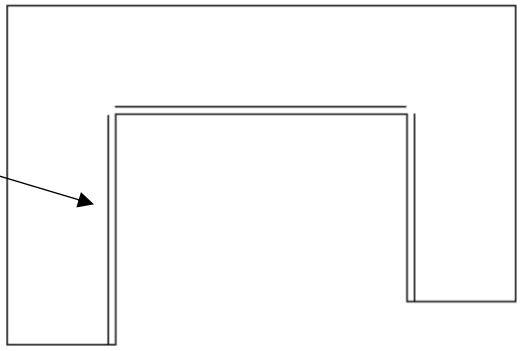


EX 13

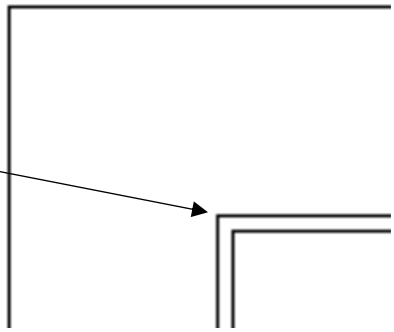
(25.5” dimension is typical in most counter applications)

Drawing a Euro Seam on an inside corner

1. **Offset** interior edges to the inside of the top typically 1.75”. *(This value may vary depending on your application).*



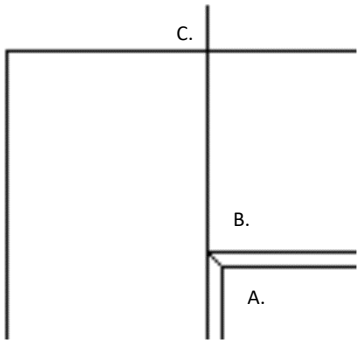
2. Use **Fillet** with the radius set to 0 to connect the corners of the offset lines if they are not connected already



3. Draw a **Line** representing the seam. Snap the **Line** at the inside corner *(A)* of the countertop (**End of – F6**).

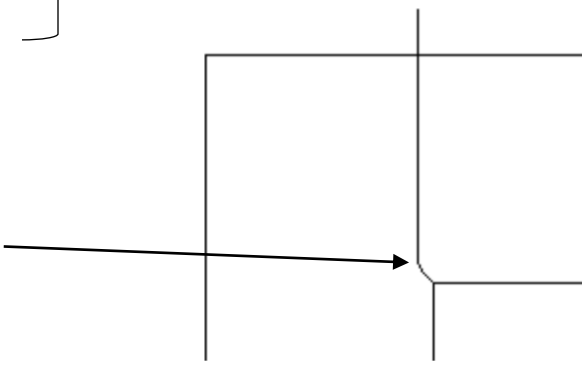
4. Snap to the inside corner *(B)* of the offset geometry (**End of – F6**).

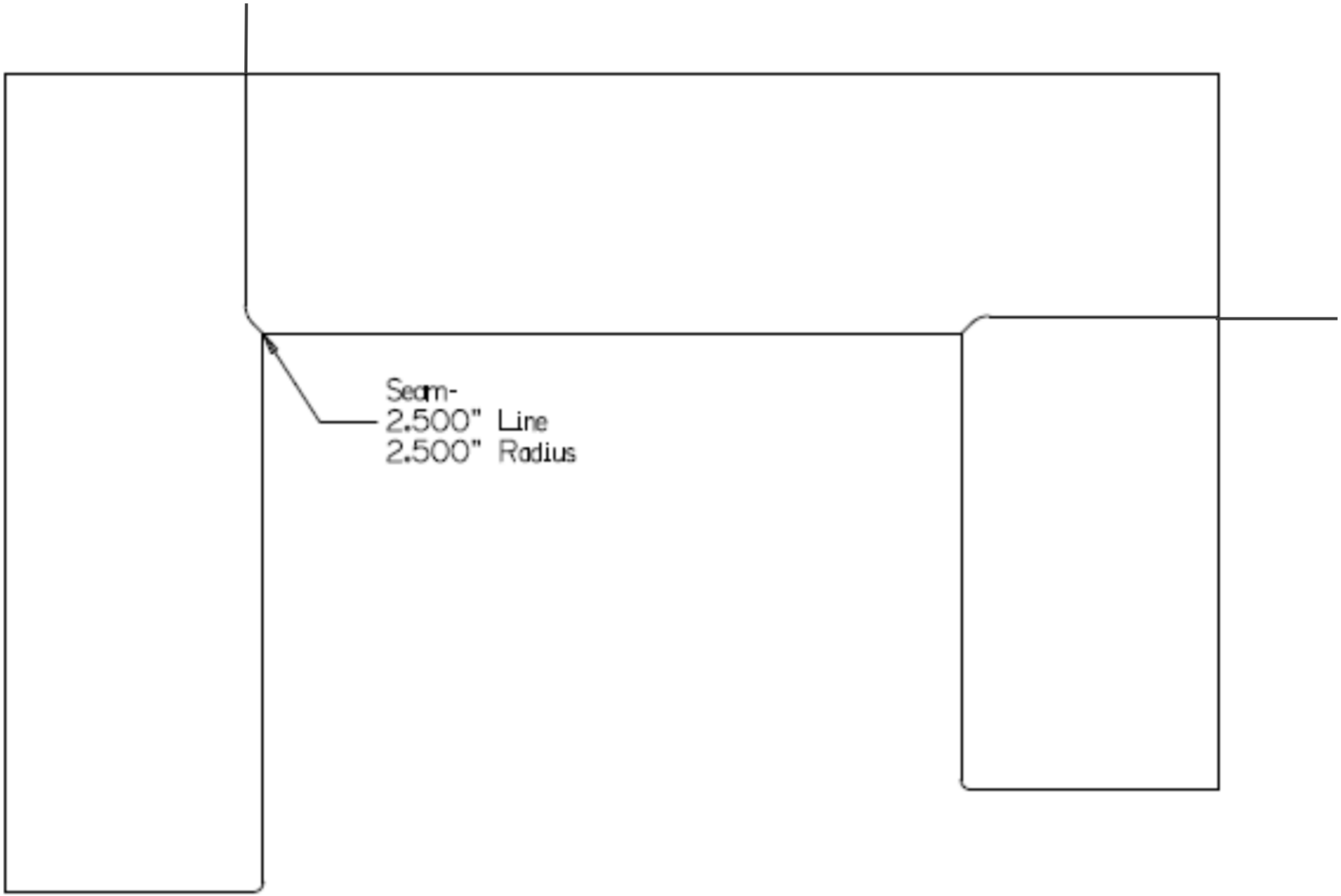
5. Then snap the line (**Perpendicular – F11**, or **Parallel – F12**) to the appropriate <u>offset</u> edge and drag it out <u>past</u> the Perimeter *(C)*.



6. **Delete** the offset lines.

7. **Fillet** the inside corner on the **seam**. Commonly a 2.5” radius. *(This value may vary depending on your application).*





EX 14

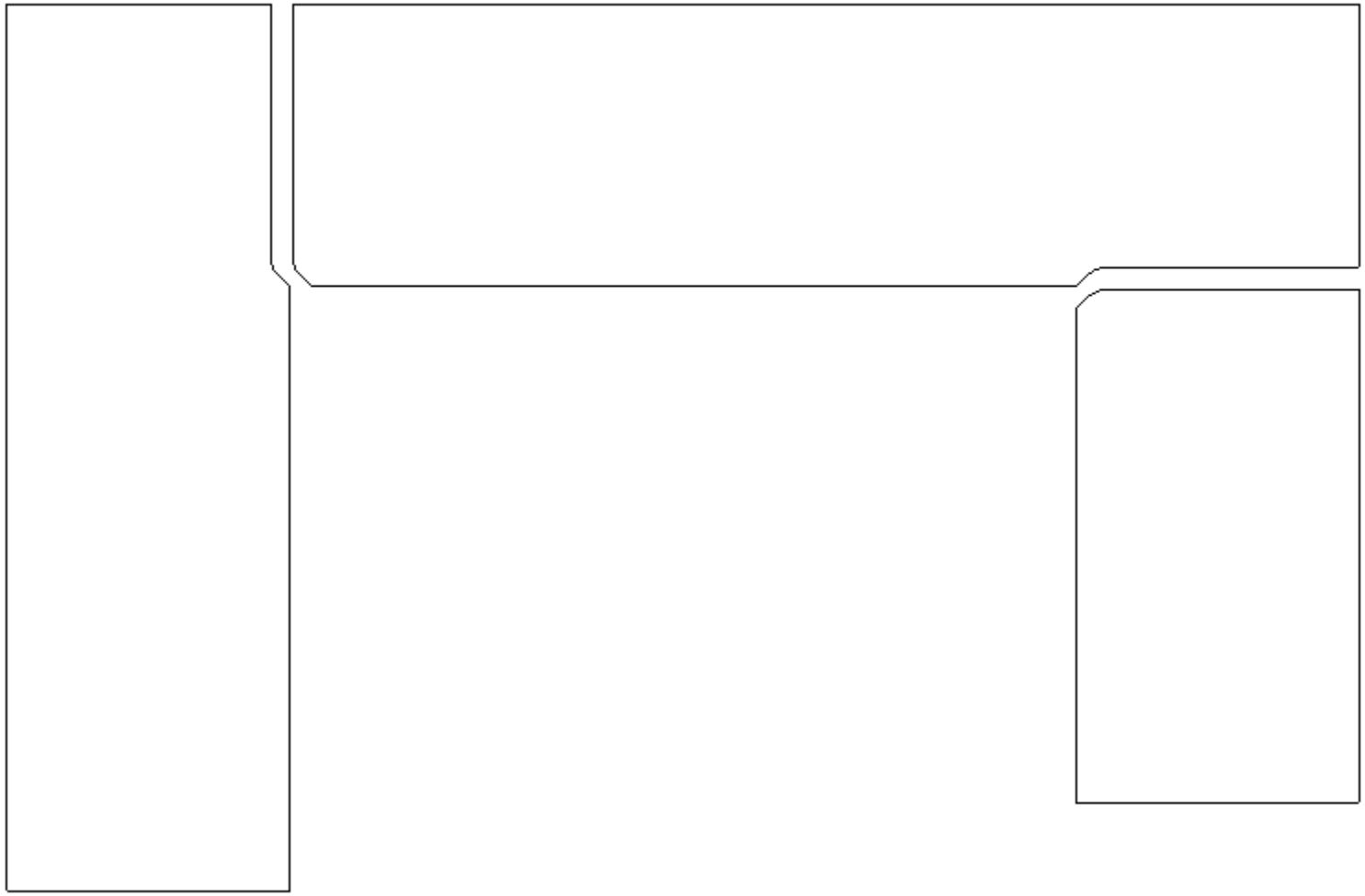
Separating a Countertop into parts

Using the **Fabricate Cut** feature in Alphacam is an easy way to separate countertops into manageable pieces.

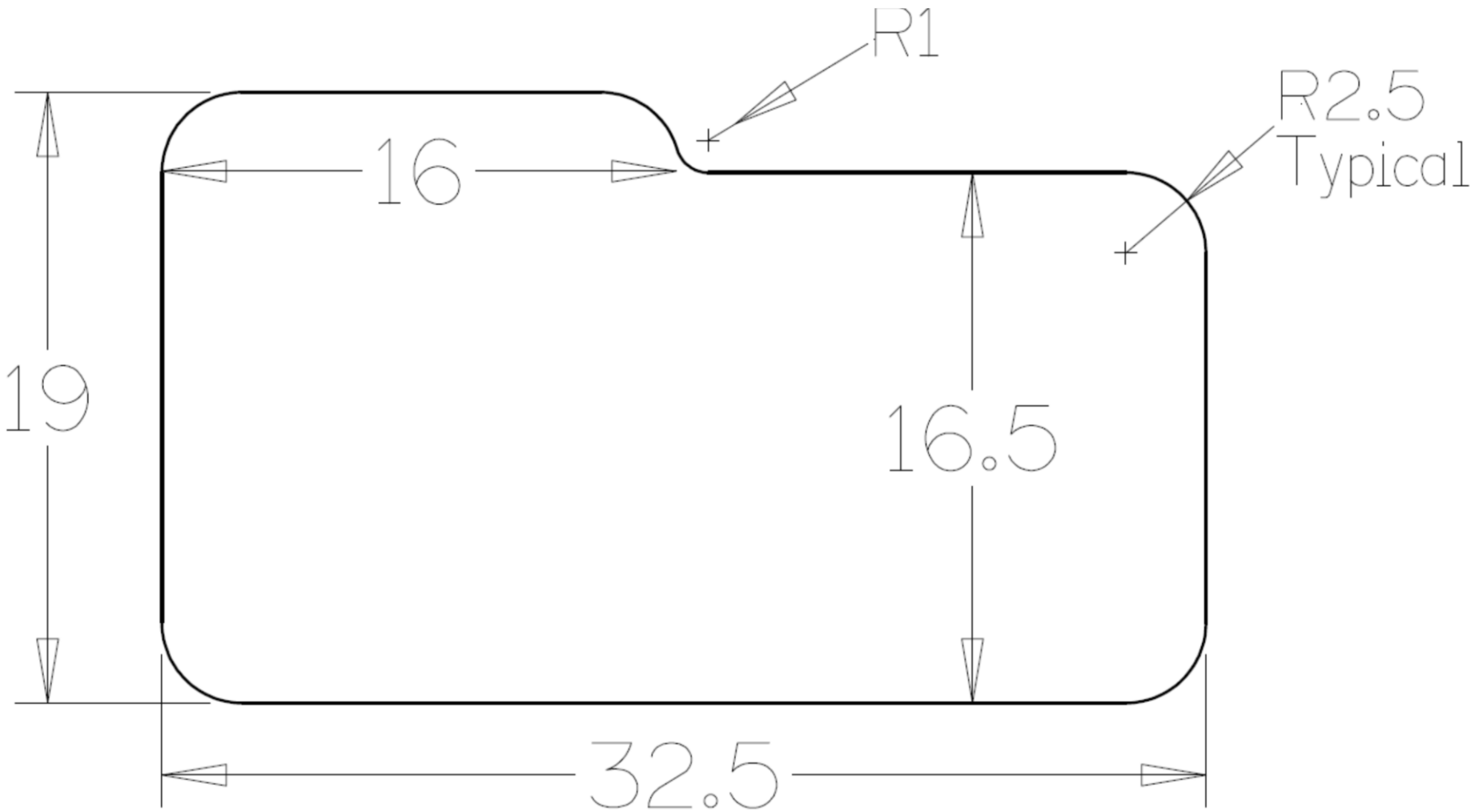
1. Draw and/or fix the countertop layout.
2. Edit- **Join** the entire perimeter of the countertop. **DO NOT** window everything, (seams and countertops) or they may all get joined together.
3. Draw and/or ensure that all seams are fully extended to (or past) the perimeter.
4. Edit- **Join** all lines and arcs making up the seams are. **DO NOT** window everything, (seams and countertops) or they may all get joined together.
5. Choose Edit- **Cut** from the Fabrication section in the Edit Tab.
6. When prompted to *"**select cutting geometries**"*, pick all the seams and then finish. (The seams will now change color)
7. When prompted to *"**select geometries to cut**"*, pick the perimeters of all countertops you are cutting, and then finish. This will end the command.
8. Edit- **Delete** the original seam geometry. If the original seam geometry was not trimmed flush to the edge, you can easily select it using the mouse pointer.

Notes:

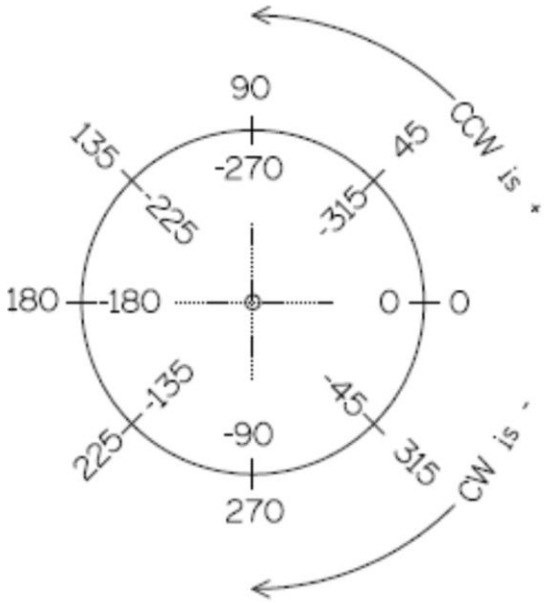
Separate into parts Edit – Fabricate - Cut



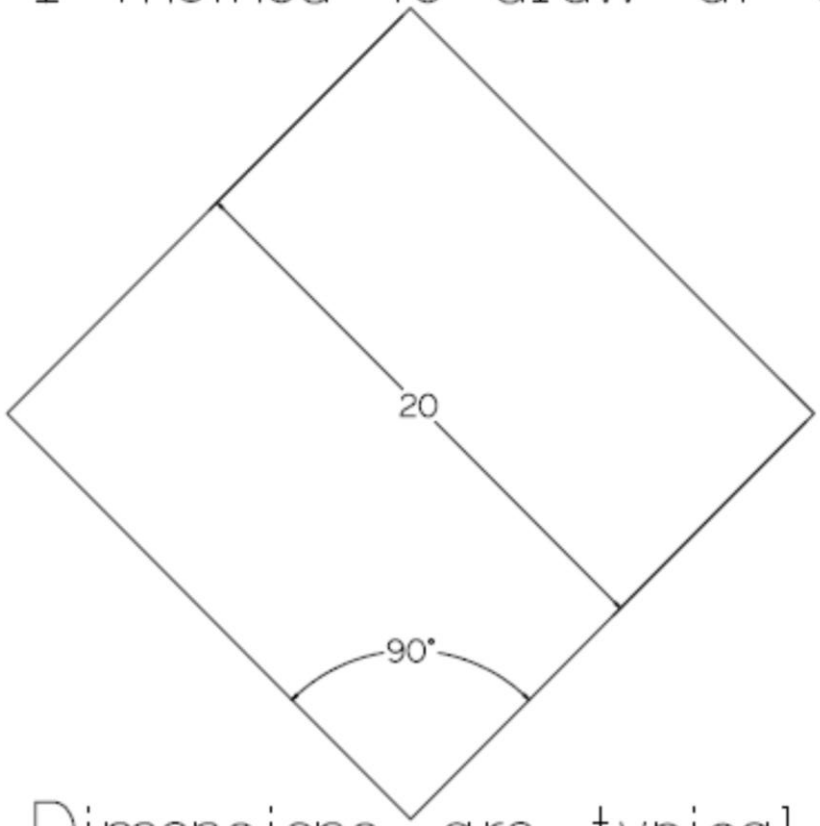
Save into your folder as Kitchen1



Save completed sink in your folder as SinkDraw.

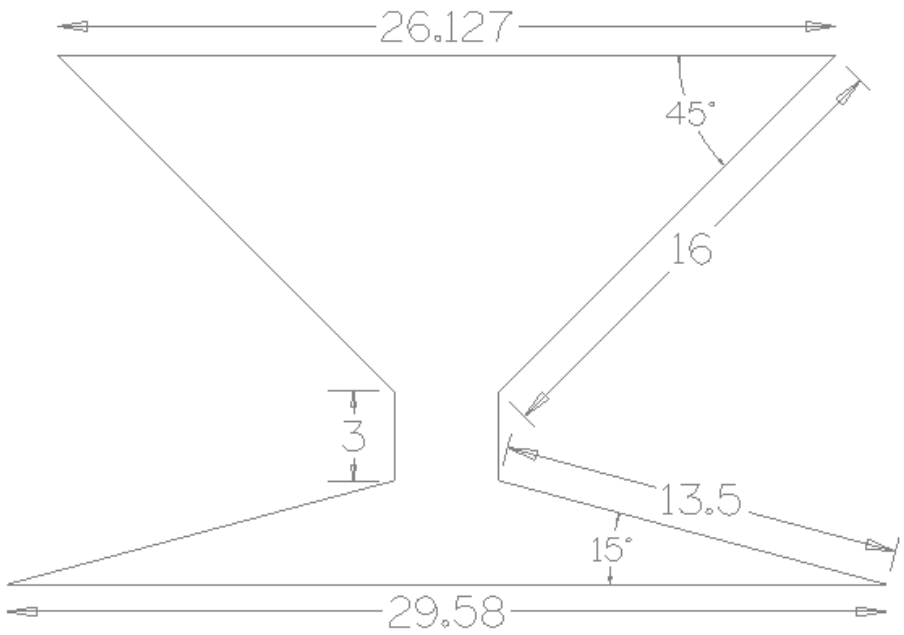


Use F1 method to draw at angles



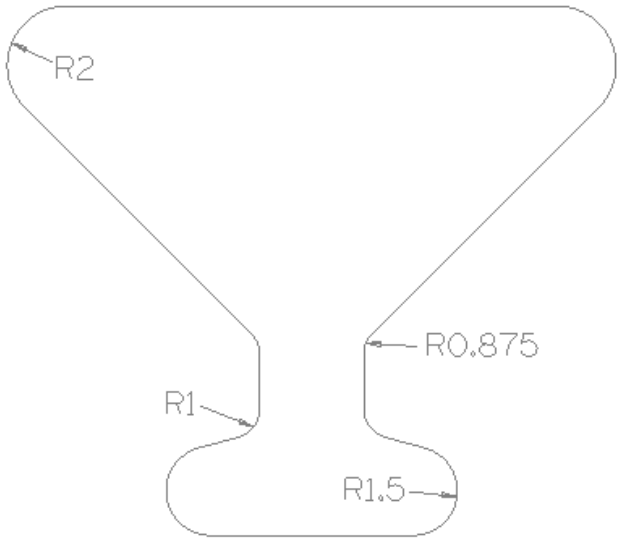
Dimensions are typical

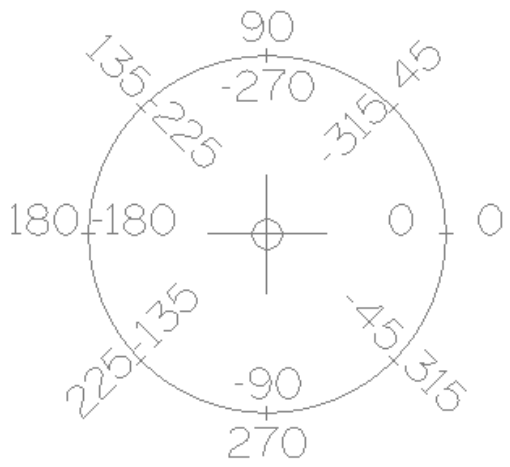
Draw lines using the F1 angle method

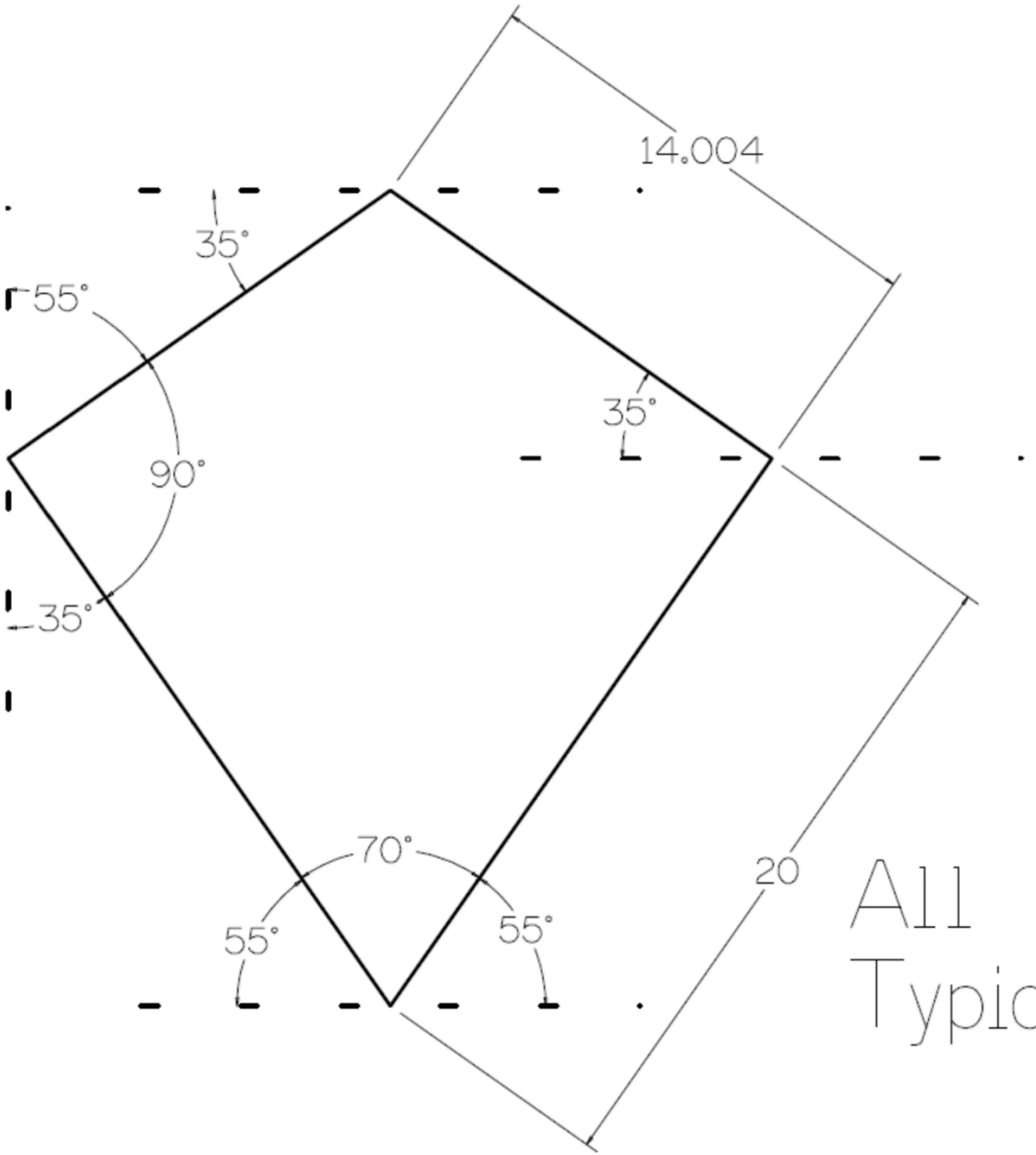


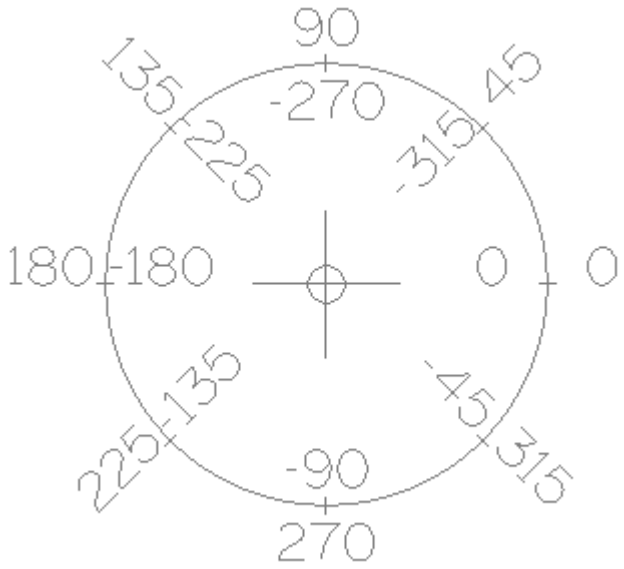
All dimensions are typical.

Complete the drawing using the Fillet

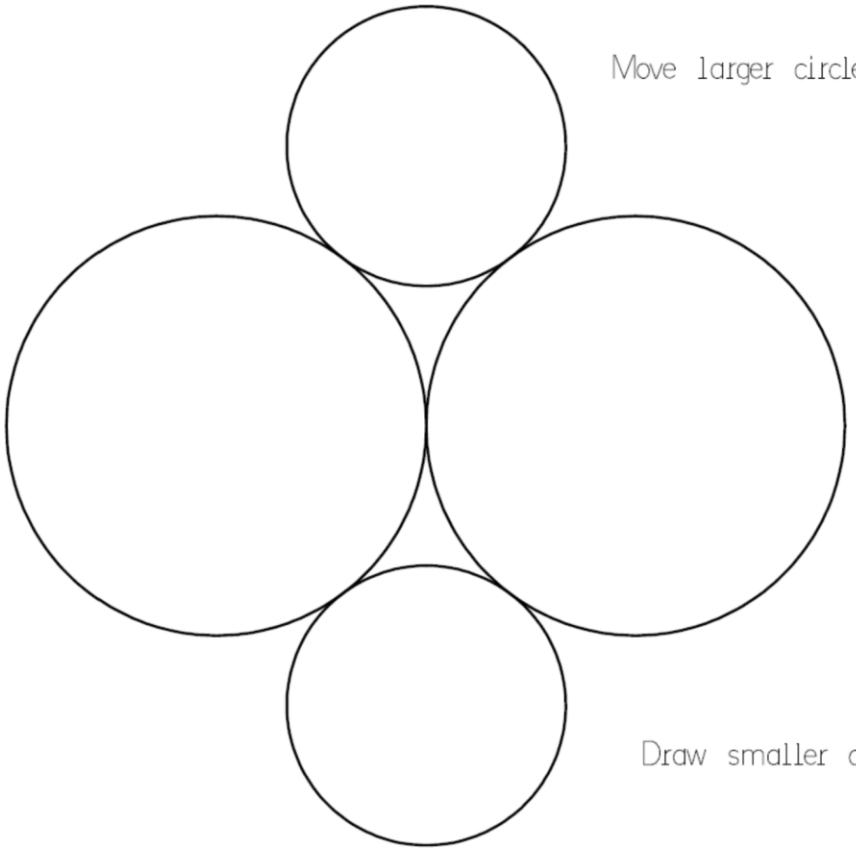




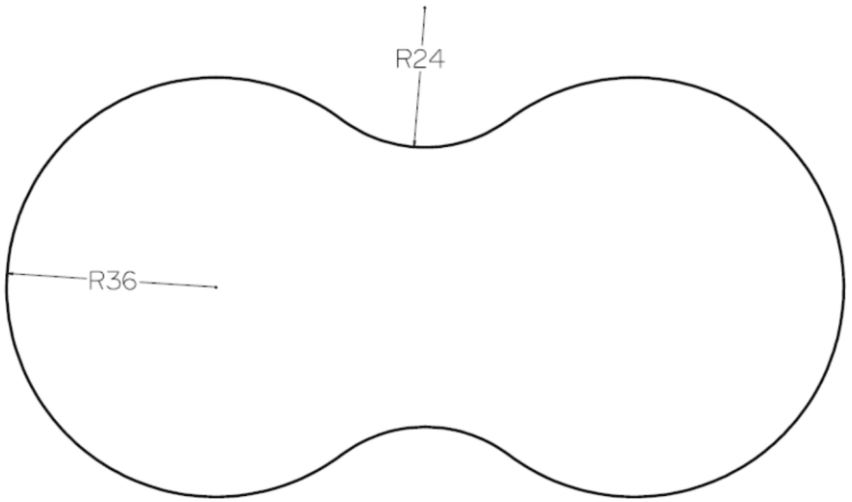




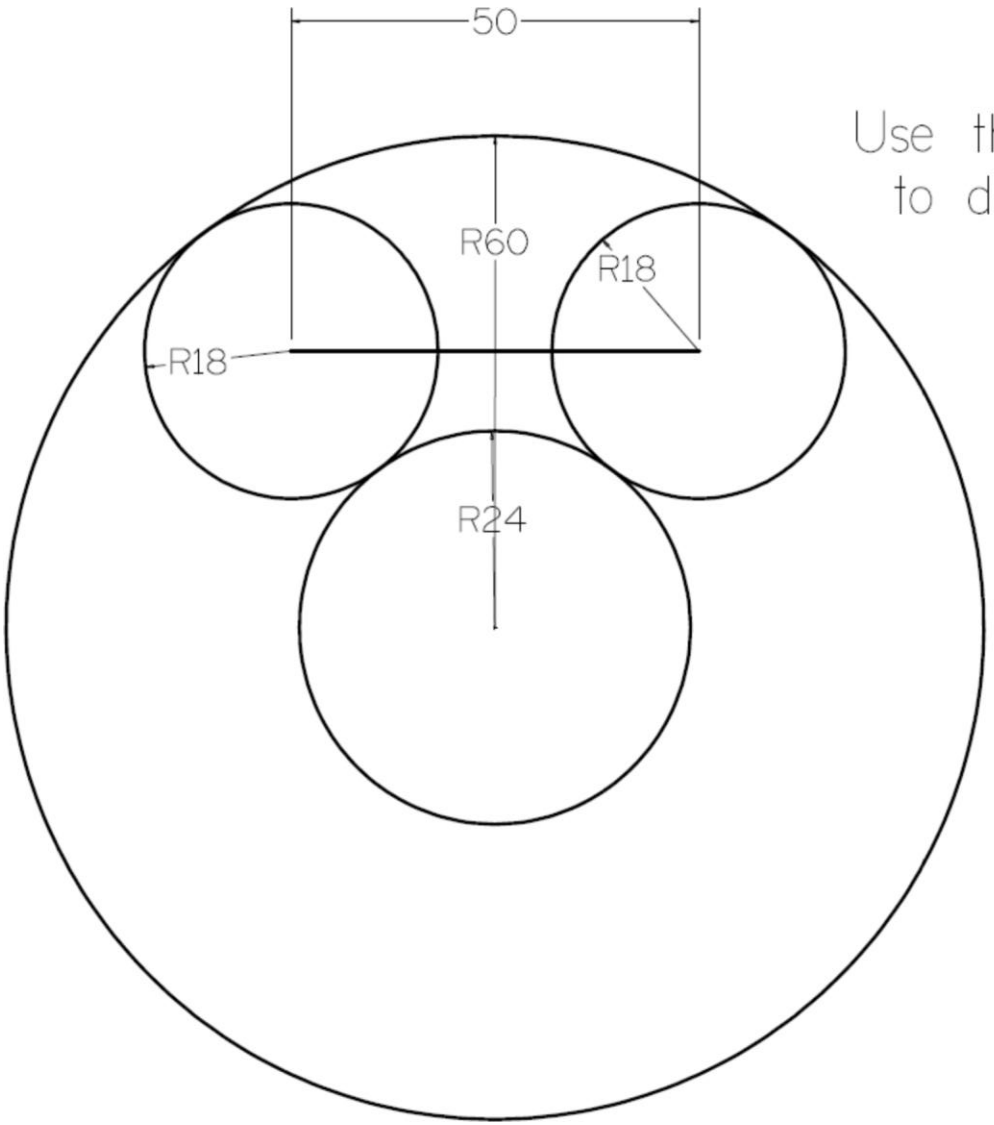
All dimensions are Typical.



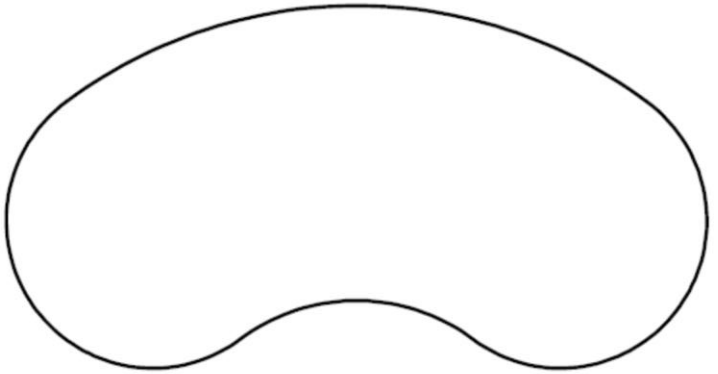
Move larger circles to each other with Quadrant snap

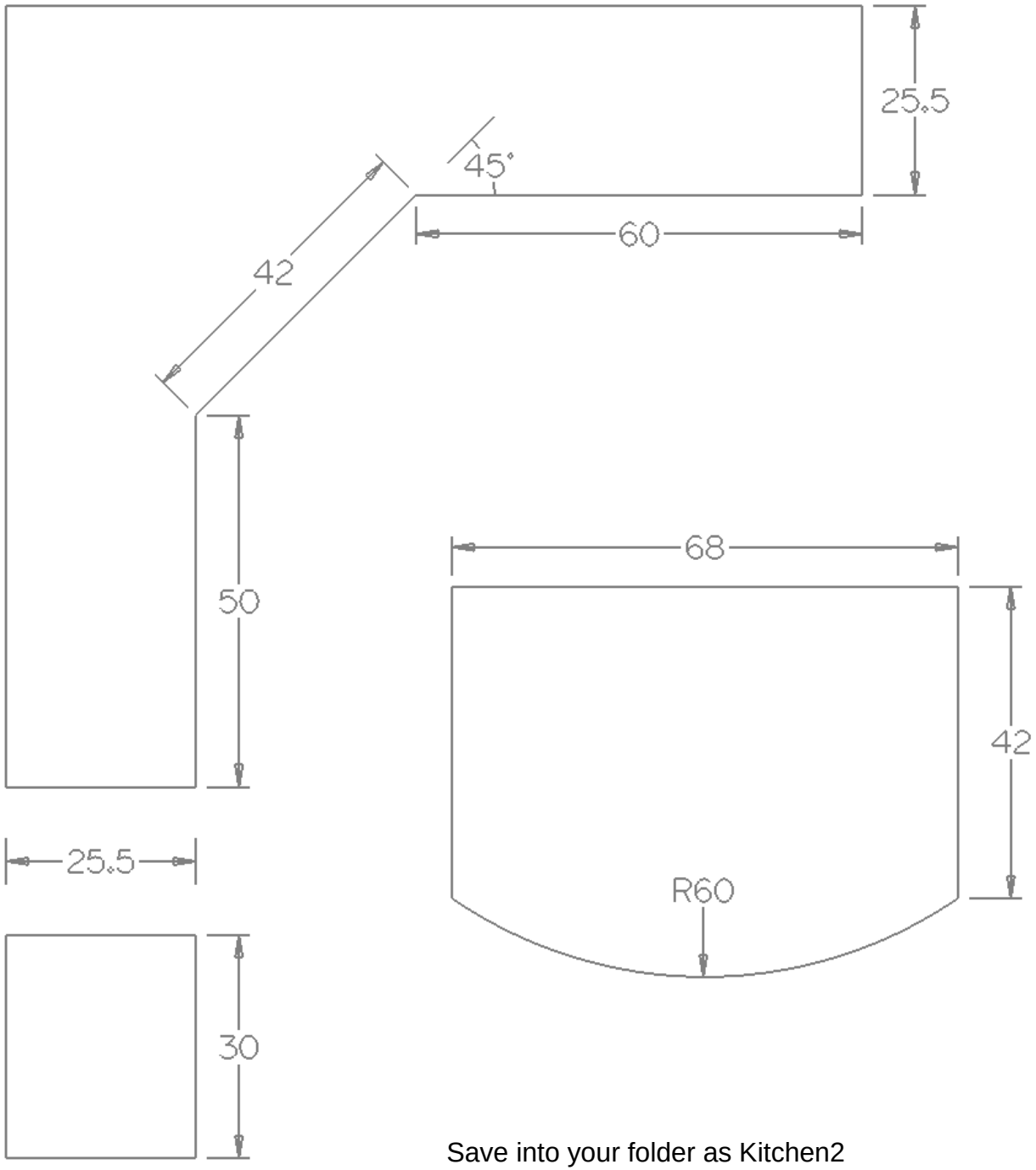


Draw smaller circles with tangent and known radius option

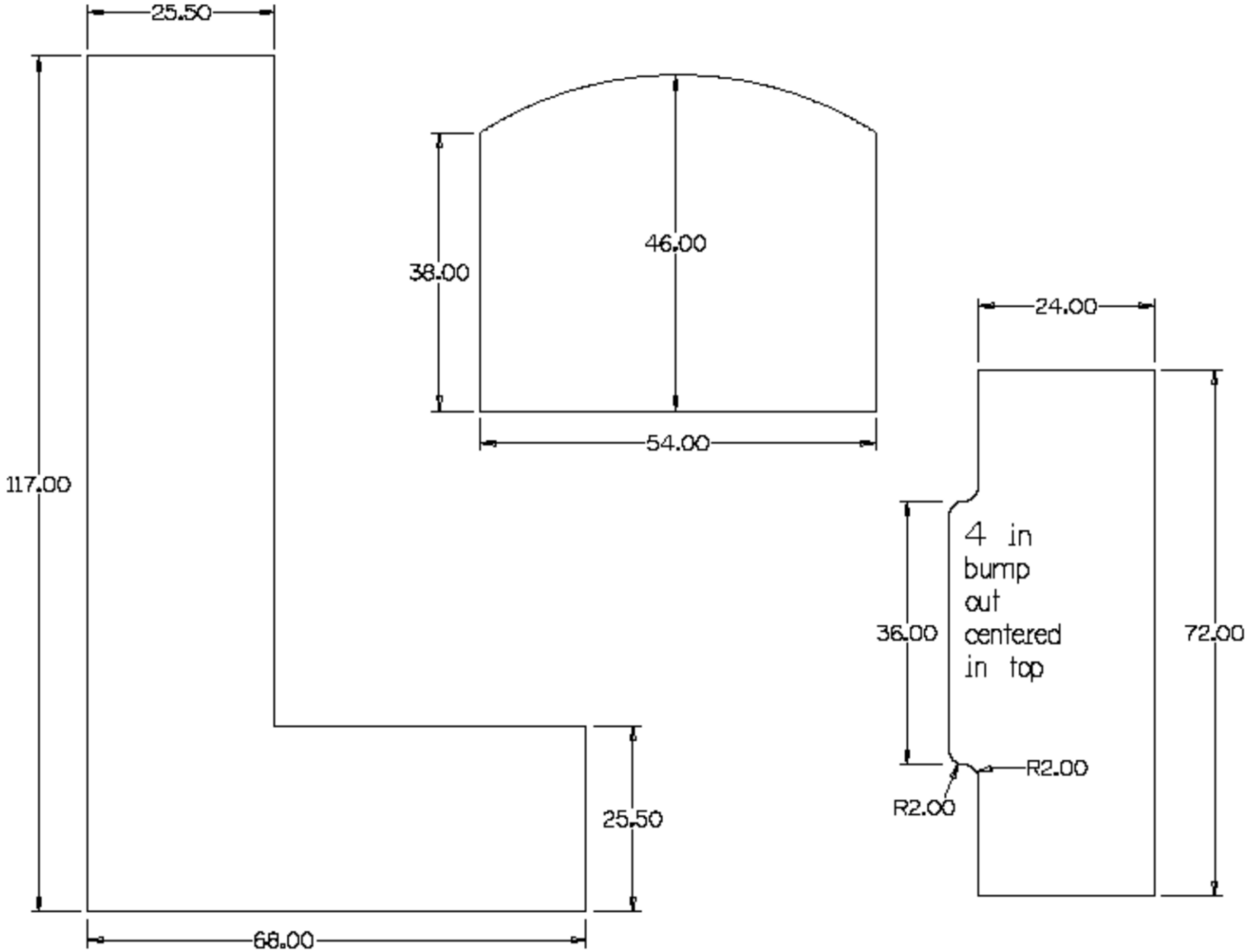


Use the Tangent and known radius option to draw the two larger circles.

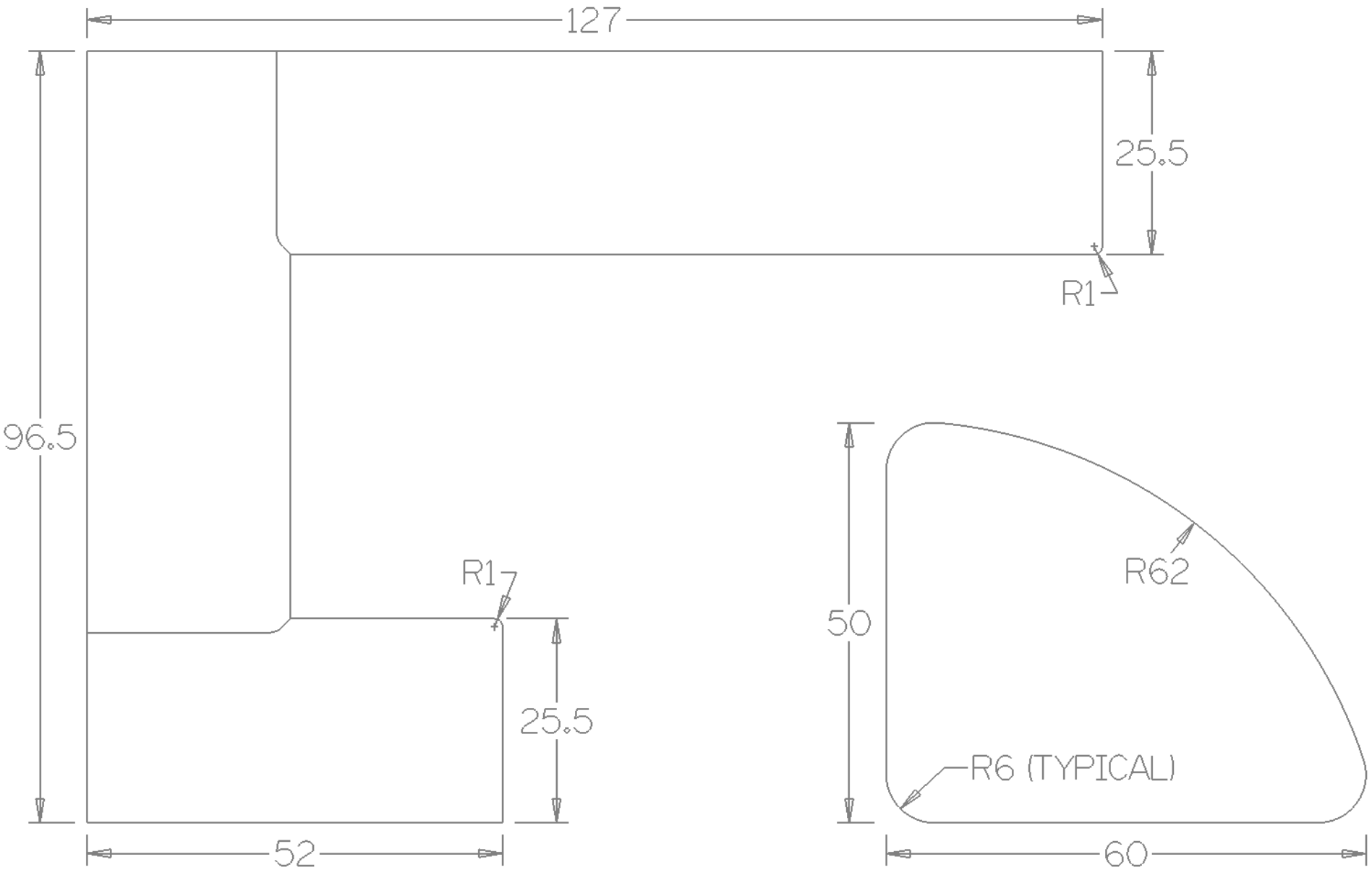




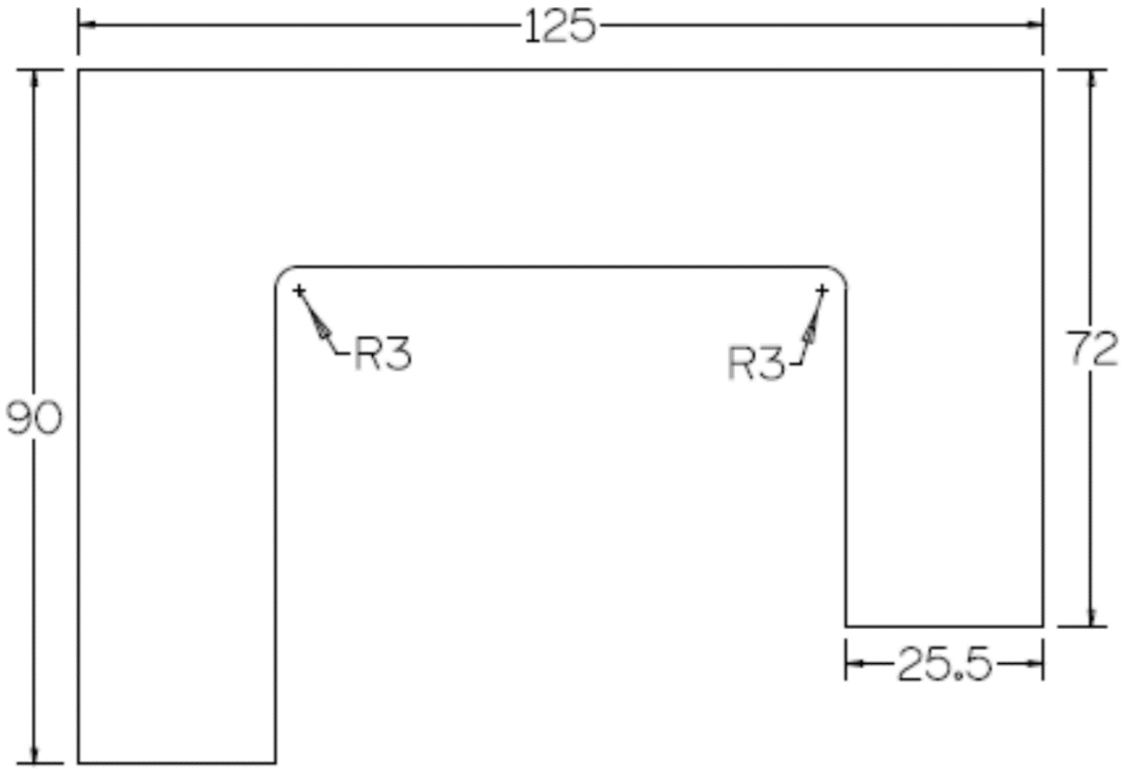
Save into your folder as Kitchen2

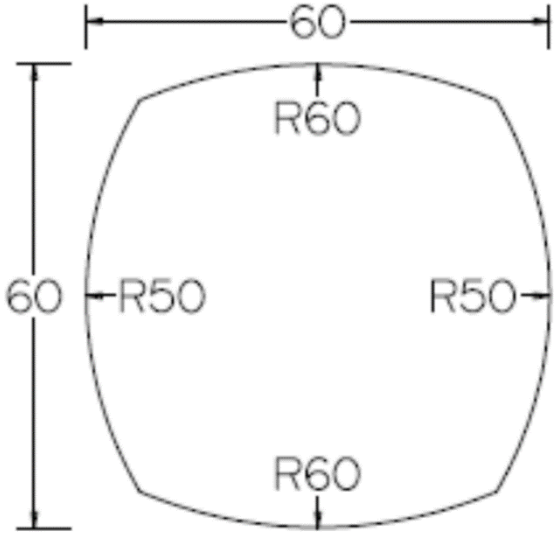


Save into your folder as Kitchen3

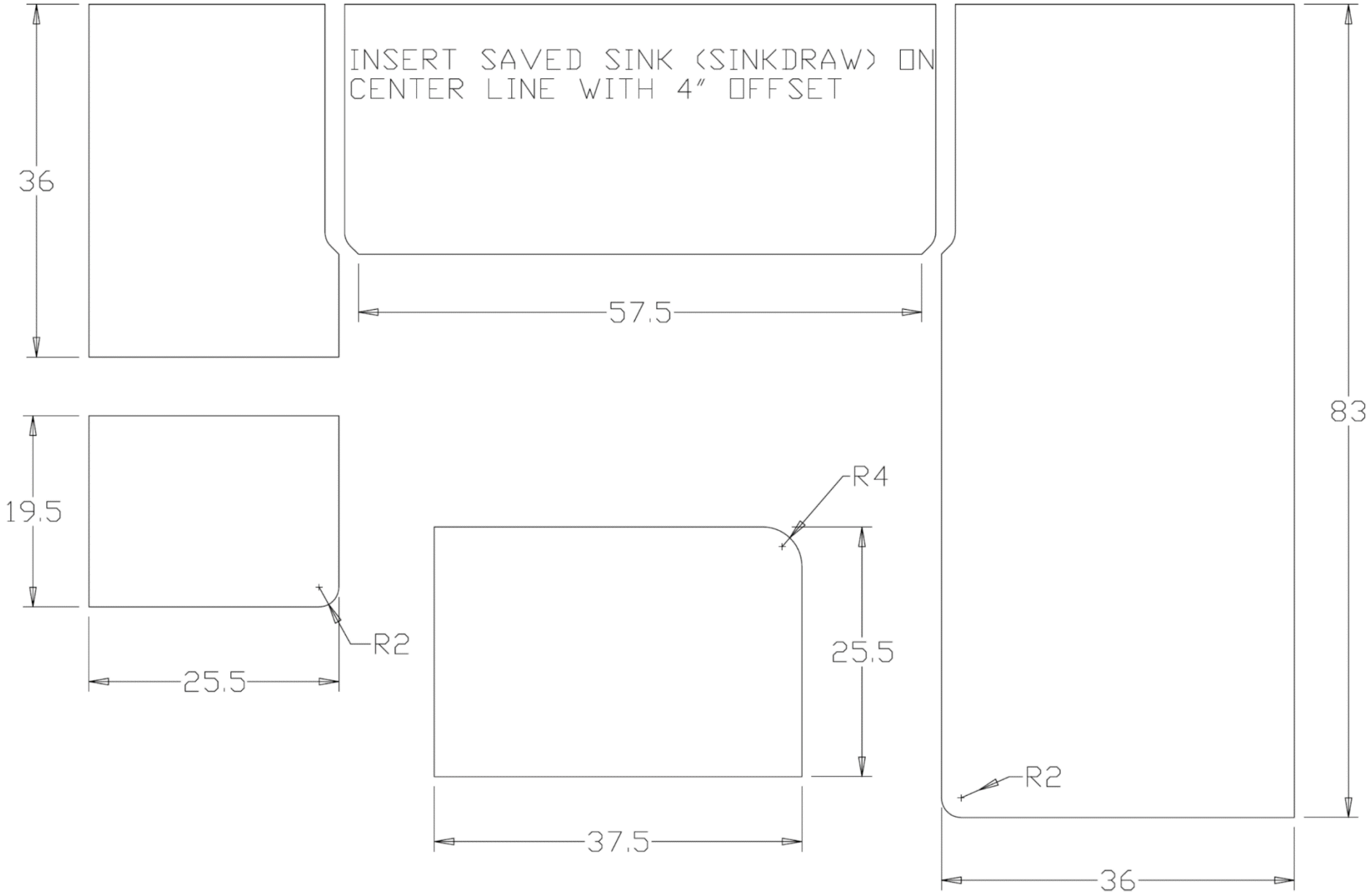


Save into your folder as Kitchen4





Save into your folder as Kitchen5



Save into your folder as Kitchen6

Section 2

Sink & Countertop Cleanup Procedure

Videos for this section can be found on our website's service and training guides.
https://www.parkindustries.com/stone/service-support/guides/

Suggested videos/guides include

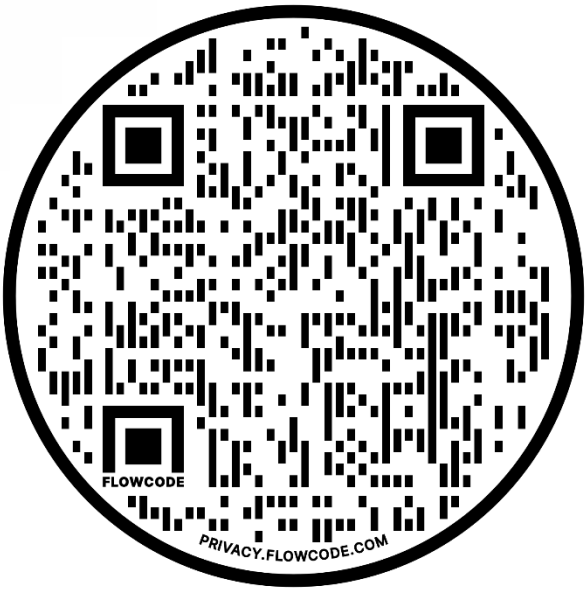
EDITING SINKS USING ALPHACAM (VIDEO)

CLEANING UP A DIGITAL COUNTERTOP TEMPLATE (VIDEO)

Other suggested videos

EXECUTING ADVANCED SINK ALIGNMENT (VIDEO)

EXECUTING ADVANCED ROTATE- ALPHACAM (VIDEO)



SERVICE & TRAINING GUIDES

Sink Editing Procedure

1. File – **New** Alphacam Drawing.
2. Home – **Input Cad**, search for desired sink .dxf file to open in Alphacam.
3. Edit – **Keep** Select items (sink outline and faucet geometries) then Finish.

   NOTE: If no **Keep** command is available be sure to turn on: Add-Ins/Macros-**Add-Ins**, expand Edit, select Keep Selected Items.
4. Edit – **Explode** Select all geometries and then click Finish.

   The white 'X's show the start points of the lines and arcs on the sink. If there are many white 'X's along the sink ('X's appear to be stacked on top of each other when the sink is viewed full screen) then the sink was drawn with splines and needs to be converted to longer length lines and arcs (see step 7). If there are minimal white 'X's, or the sink drawing already has brown spline elements, then skip step 7 A). Be sure to have the **Show Breaks** turned on-**View-Display Options-Show Breaks**
5. Edit – **Change** (From- Old Layer To- Geometry) Don't forget to select all geometry and then click Finish!
6. Geometry – Special Functions – **Common Line Removal.**

   Use a selection window to select all geometry and then click Finish. Choose "Yes" to Join Resulting Geometries. Zoom in on the 1 white 'X' to be sure there is no gap or overlap. If there is more than 1 white 'X' on a geometry, or there is a gap, overlap, or extra geometry on the outline it needs to be fixed before proceeding. (Use Delete, Trim, Blend, Fillet, Line, Arc, etc. to fix the problem then be sure to **Join** it all together.
7. Converting Splines to longer length lines and arcs- ***(Only if Needed)***.

   A) Geometry-Splines-**Create Spline**- Set the tolerance to *.010* and check the box to *Delete Original*. Press OK, select only the sink geometry and Finish. The geometry will turn all brown. If the geometry already has brown spline elements when opened, skip this step, and go directly to step B - **Convert Spline to Lines/Arcs**.

   B) Geometry-Splines-**Convert Spline to Lines/Arcs**- Set the tolerance to *.010* and check the box to *Delete Original* and *Join Resulting Lines/Arcs*. Press OK, select the spline, and then Finish. **Explode** the geometry to verify the longer length lines and arcs (the white 'X's will be spaced further apart), and then **Join** the geometry back together.

8. Geometry – Special Geometries – **Enclosing Rectangle.**  Select only the sink geometry and then Finish.
9. CAD – **Distance/Angle**  or CAD- **DIMENSION**  Measure the Enclosing Rectangle to find the size of the sink. Measure along both the X and Y axis. If using DISTANCE ANGLE, measure from the bottom left corner of the rectangle to the top right corner of the rectangle to get both axis. Be sure to use the F6 snap function when selecting both end points.
10. Edit – **Scale**  *(This step only needs to be done if the sink was originally drawn in millimeters and needs to be converted back to inches.)*

    Select all geometries and then Finish. Left click in the middle space of the sink for your Base Point. Type in a Scale Factor of 1/25.4 (0.03937) for the X and Y values. (Imperial to Metric 1 inch = 25.4 mm) (Scale = Want / Have)

    Or: Measure the physical sink or template to find the desired size in inches. Write or record those numbers down. Then measure the sink in Alphacam using the **Distance/Angle** command as instructed in the previous step (9) and record the values for X and Y. Choose the **Scale** command and select all geometries and then finish. Left click in the middle of the sink for your Base Point. Divide the sink bowl opening in inches by the measured sink bowl opening in Alphacam for the Scale Factor in X. Repeat division process for the Scale Factor in Y. Measure the sink again in Alphacam after scaling to be sure it was scaled correctly.
11. Edit – **Offset**  *(Only if the sink needs to be offset for a reveal or overhang)*

    When using offset, enter the value to offset and select Geometry and Delete Original in the options. Offset the sink geometry and also the Enclosing Rectangle before exiting the command.
12. Edit – **Move**  *(Change insertion Base Point by moving sink center to X0,Y0)*

    Select all the geometry, then Finish. Choose the Base Point by using the Midpoint Snap (F7) and selecting the bottom line of the Enclosing Rectangle. Enter 0 for the X coordinate and 0 for the Y coordinate and press OK.
13. Edit – **Delete**  Delete the Enclosing Rectangle from the drawing. *(If Desired)*
14. File – **Save As**  Save the file to a desired location *(Edited Sinks folder)*.

Sink Editing Fast Track Checklist

1. File – **New**  Alphacam Drawing.
2. Home – **Input Cad**  Search for correct sink file.
3. Edit – **Keep**  Selected Items.
4. Edit – **Explode**  To see if sink has Splines or not.

   *Be sure **Show Breaks** option is turned on-**View-Display Options-Show Breaks**
5. Edit – **Change**  From Old Layer to Geometry Layer. Select all and Finish.
6. Geometry – Special Functions – **Common Line Removal.** 

   ***Be sure to use a selection window and select all geometry.

7. Converting **Splines**  to longer length lines and arcs- (***Only if Needed***).
   A. Geometry – Splines – **Create Spline** – Set the tolerance to *.010"* and check the box to *Delete Original*. *(Skip this step if there are already brown splines and go to step B.)*
   B. Geometry – Splines – **Convert Spline to Lines/Arcs** – Set the tolerance *to .010"* and check the box to *Delete Original* and *Join Resulting Lines/Arcs*.
   C. **Join** the sink back together if you exploded it to check it.

8. Geometry – Special Geometries – **Enclosing Rectangle.** 
9. CAD – **Distance/Angle**  or CAD- **DIMENSION**  Measure Enclosing Rectangle to confirm sink size.
10. Edit – **Scale**  **(*Only if the sink was originally drawn in millimeters).*** *Use scale factor of 1/25.4. Measure using **Distance/Angle** to confirm size is correct.
11. Edit – **Offset**  The sink opening *if needed* for an overhang or reveal.
12. Edit – **Move**  The sink and Enclosing Rectangle to the X0, Y0 location.
13. Edit – **Delete**  The Enclosing Rectangle.
14. File – **Save As**  "Sink Name" into your "Edited Sinks" folder.

Template Cleanup Procedure

How to get a DXF file created by a digital template system into Alphacam and prepared for programming.

1. File- **New** Alphacam Stone Drawing.
2. Home- **Input Cad** Search for desired template (.dxf -.dwg -.iges) file and bring it into Alphacam. Always choose “Show colors of lines and arcs” and “Maintain Layer Visibility”. Some drawings may have reference points. You may reference these points with a circle, but usually just choose “Ignore points”.
3. Park Machine**- Add Labels** Choose a label name and select each edge (as prompted) to label if not done during templating.
4. Edit- **Keep** or **Delete** Select all elements necessary to complete the countertop outline layout for programming. You may also want to keep any references for features such as: Sinks, Cooktops, Seams, Edge finish, or Notes. If no Keep command is available, be sure to turn it on under “Add-Ins”.
5. Edit**- Explode** The white ‘X’s show the end points of the lines and arcs on the countertop. Be sure to turn on **Show Breaks** **(**View-Display Options)
6. Edit**- Delete** any extra lines or arcs that are not needed to make up the outline of the countertop.
7. Edit- **Change** Choose Old Layer in the “From” area, and Geometry in the “To” area. Do not forget to select all the components, and then finish to complete.
8. Geometry-Special functions- **Common line removal** Use a selection window around all countertop(s), then finish. Choose ***Yes*** to “Join resulting geometries?” More than one white X on each geometry indicates that there is a gap, overlap, or extra geometry. Use **Blend, Fillet, Trim, Delete** etc. or draw a new **Line** or **Arc** to close the geometry. Once fixed make sure the geometry is Joined before proceeding.

   To be sure you have closed the Geometry, use View-Display Options- Geometry- **Show Open Geometry or zoom in on the 1 white ‘X’ to be sure there is no gap or overlap on this location.*
9. If Needed
   1. Home- **Insert** any needed sinks (Pg. 44- Inserting Sinks), faucet holes, or cooktops.
   2. Edit- **Cut** (Pg. 26) the pieces apart at known seam locations (Pg. 24- Drawing Euro Seams).
   3. Backsplash and Aprons (if using Slabsmith draw aprons as backsplash)
10. File- **Save As** into the correct job folder for that customer and re-name if needed. File is now ready for the Machine Programming Checklist.

Inserting a Clean Sink File

Create your parts or open an existing drawing in Alphacam.

1. **Offset** the front edge to the inside of your part to create the sink *Setback line* where the front edge of the sink will be placed *(commonly 4")* using the *Line / Arc* option in the **Offset** command.
2. Draw a **Line** or use **Offset** to create a sink *Centerline* that will be used to accurately place the sink in the correct spot on your part.
3. Go to the **Home** tab and choose **Insert**. You will then browse on your computer for the correct sink file that you desire and open it. *(This would be a sink drawing that you have previously cleaned up and saved in Alphacam. See sink editing procedure pg. 40-43.)*
4. If you saved your sink correctly at the 0,0 location, you will be holding the sink at the base centerline with your curser. Snap it to the **Intersection (F9)** of the *Setback line* and the *Centerline* you previously created.
5. If you are unsure of the angle of the countertop's front edge, measure the *Angle* of the *Setback line* using the **Distance / Angle** function in Alphacam. Use your **End Point (F6)** snap key and snap each end of the line measuring in a positive direction. If the angle measures 0 or 360 degrees, then you do not need to rotate the sink as long as the sink has the correct orientation inside your part. If the angle is neither of those degree values, or if the sink is not orientated correctly you will need to **Rotate** the sink so that it becomes *Parallel* with the front edge. You will need to proceed to step 6.
6. *(Perform these steps to Rotate the sink only if needed)* Choose the **Rotate** command from the *Edit* tab and either select the *Previous* button on the bottom of the screen or select only the sink and faucet holes as needed and then *Finish*.

   When prompted to pick a *Base Point* press the **Intersection (F9)** snap key and then select the *Setback line* and the *Centerline* that were originally used to snap the sink to.
   Once the sink is pivoting on the base point you are prompted to choose the *Rotation Angle*. You will then press the **Endpoint (F6)** snap key and select the correct side of the *Setback line* to snap the sink at the correct angle and correct orientation inside your part. When prompted for *How Many Copies* select 1 and press OK, and then *Finish* the command.
7. **Delete** the *Setback line* and the sink *Centerline*.

Notes:

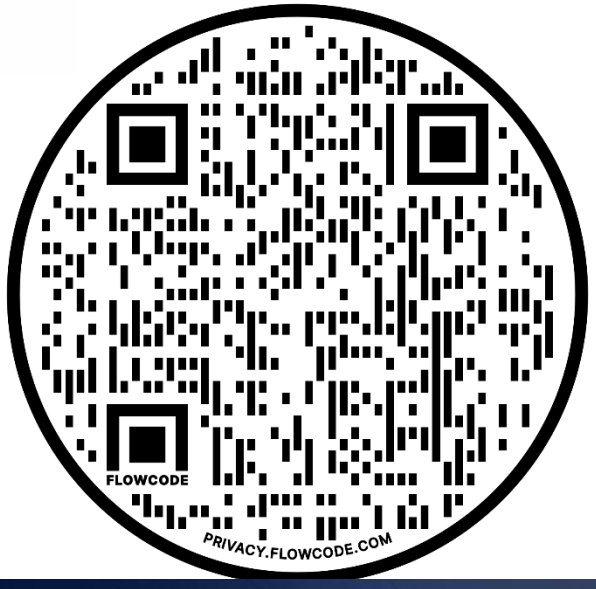
Section 3

Tool Drawing Information

Videos for this section can be found on our website's service and training guides.
https://www.parkindustries.com/stone/service-support/guides/

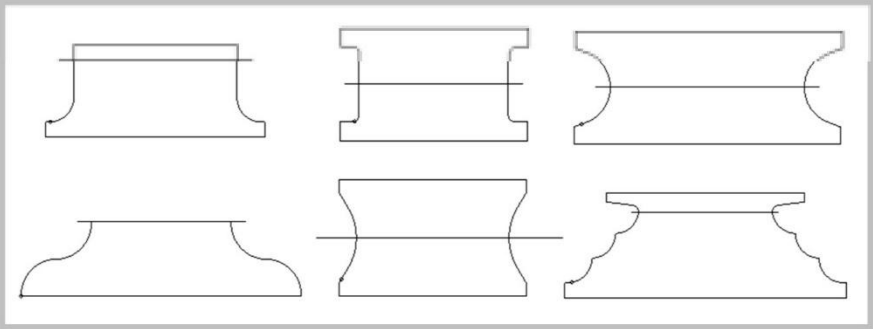
Suggested videos/guides include

CREATING TOOLS IN ALPHACAM (VIDEO)



SERVICE & TRAINING GUIDES

User Defined Tool Drawings



User Defined Tool Drawings

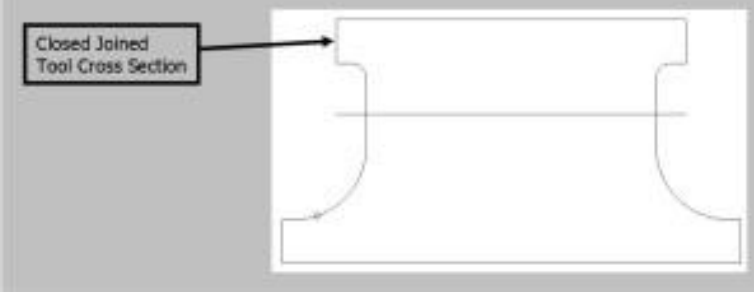
- Allow an accurate 3D simulation of parts being cut
- Used to calculate the lead in and out for each tool
- Used to take accurate measurements for ITM purposes
- Needed only when the tool profile is not flat
  - Flat profiled tools include: Core, Mill Bit, Zee Wheel, Gauge, Blade, etc.

Three Rules of User Defined Tool Drawings for Programming

- Perimeter of the tool drawing must be **joined**
- **Horizontal line** extending across the smallest diameter
- **Small circle** to define the Z0 point

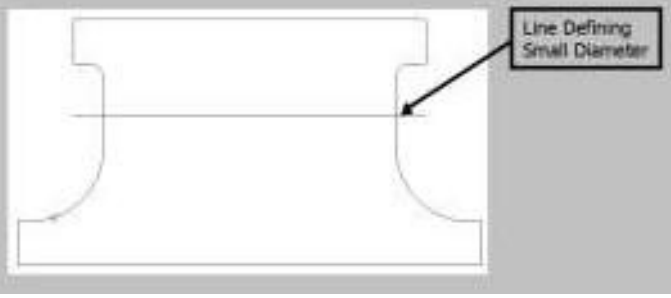
User Defined Tool Drawings

- Cross section outline of the tool
- Use a combination of caliper measurements and manufacturer supplied dimensions
- Must be JOINED!



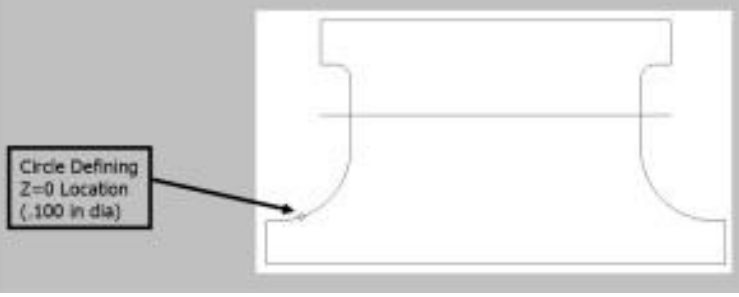
User Defined Tool Drawings

- Horizontal line
- Must cross the tools small diameter
- Must extend past the tool outline



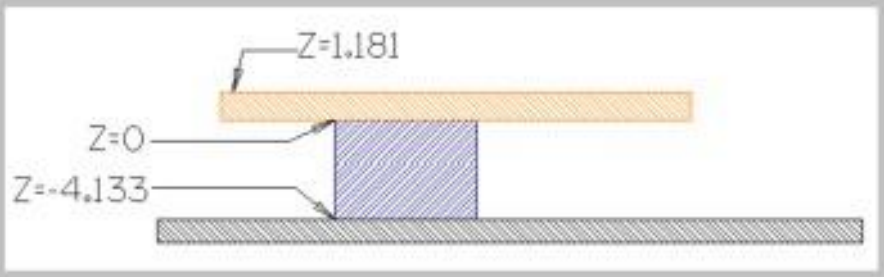
User Defined Tool Drawings

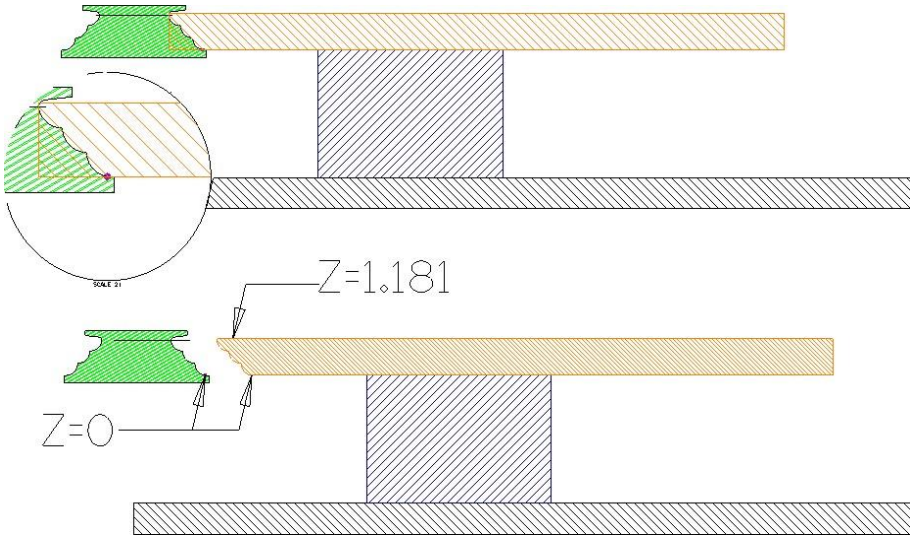
- Circle size is not important (.100 dia works well)
- Circle center must be on the tool outline
- During simulation the tool will run with the circle at Z=0 (top of pod)

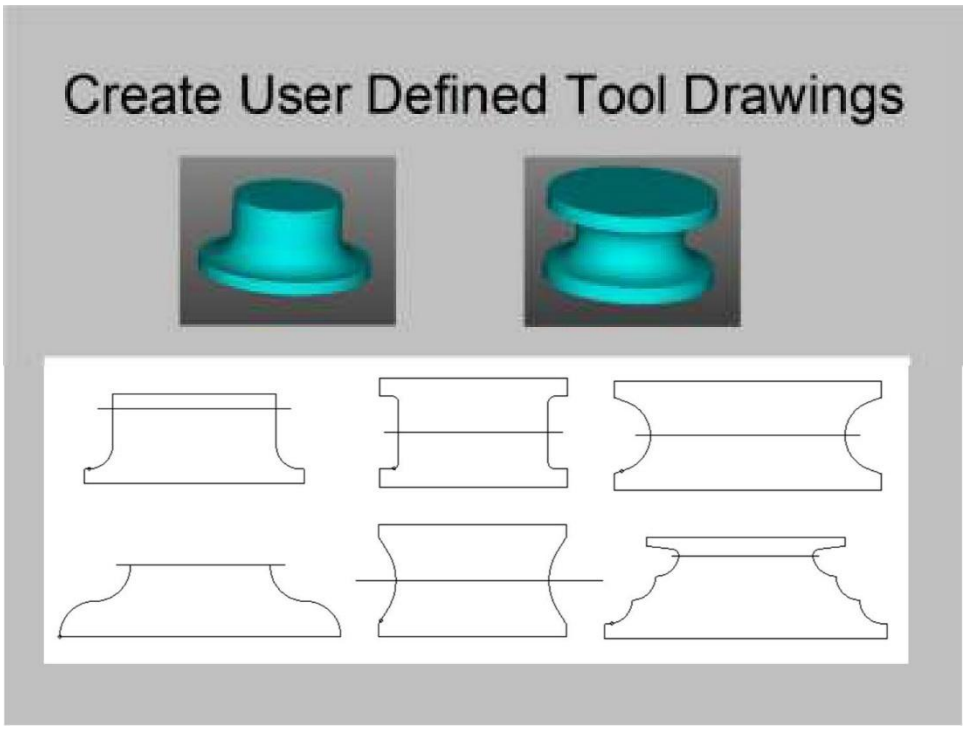


Machine Z Heights

- The top of the vacuum pod is Z=0
- Above the pod is +Z
- Below the pod is –Z



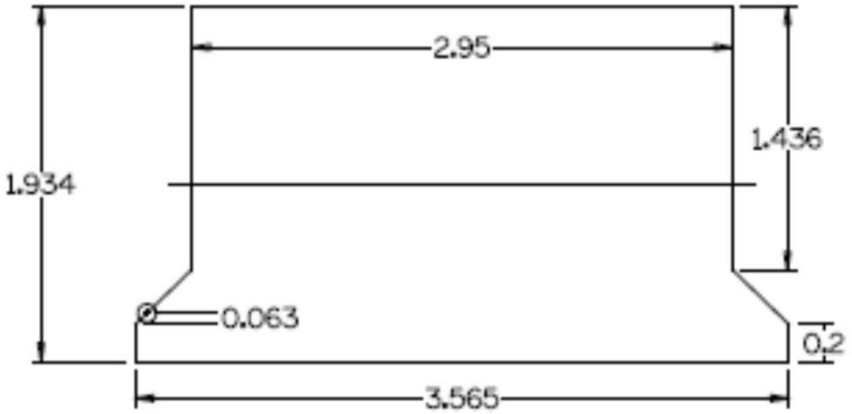




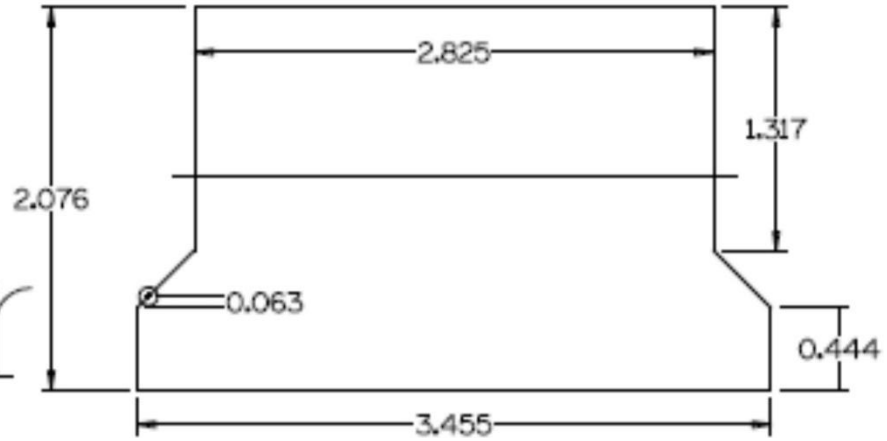
The following pages are examples to create tool drawings for programming purposes. The exact dimensions are not always available from tool manufactures. If they aren't available, the actual tool may be measured with a caliper to get the important measurements. The important measurements are the large and small diameters. These measurements and values are used in generating the Lead in and out in the programs. All other measurements are only used for "Clash Detection" with the vacuum pods in the 3D simulation.

ES-30

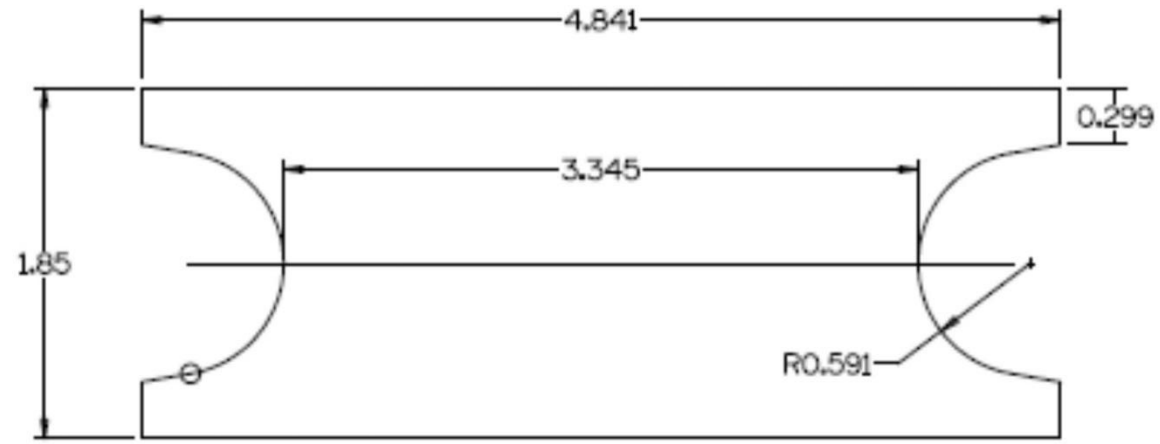
Metal



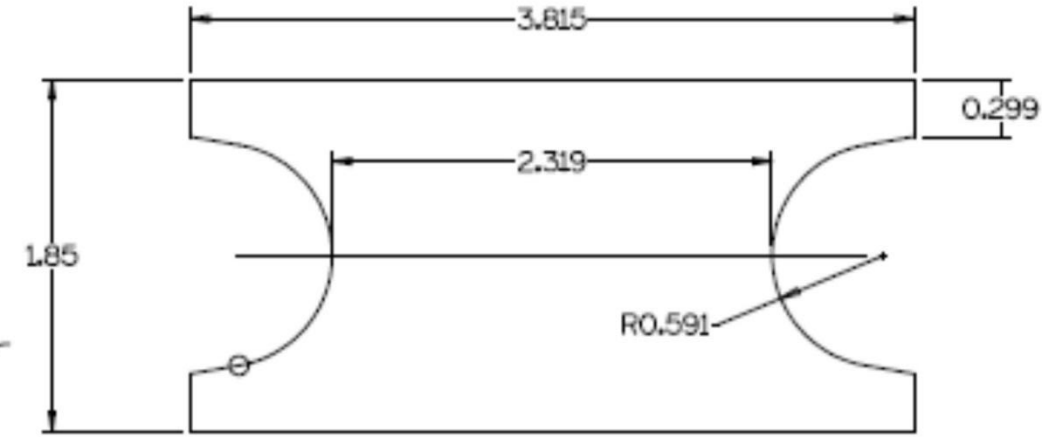
Polisher



V30 Metal

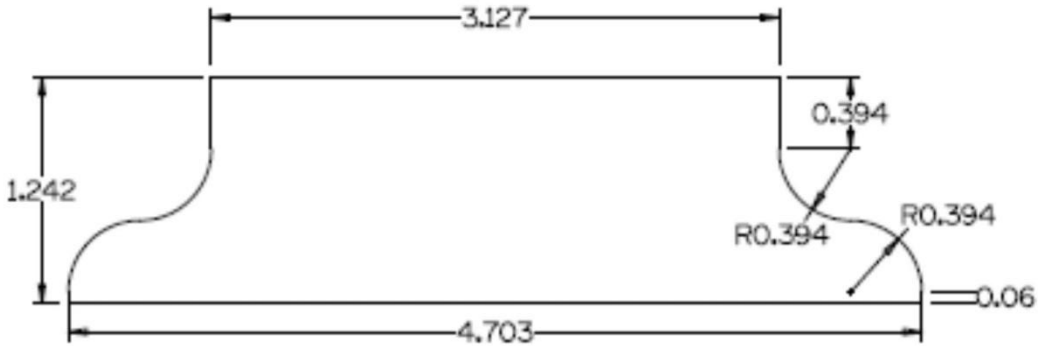


Polisher

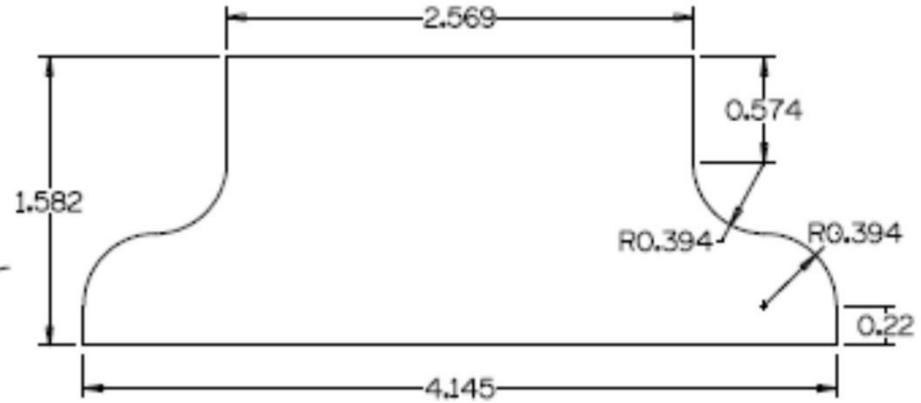


FZ-30

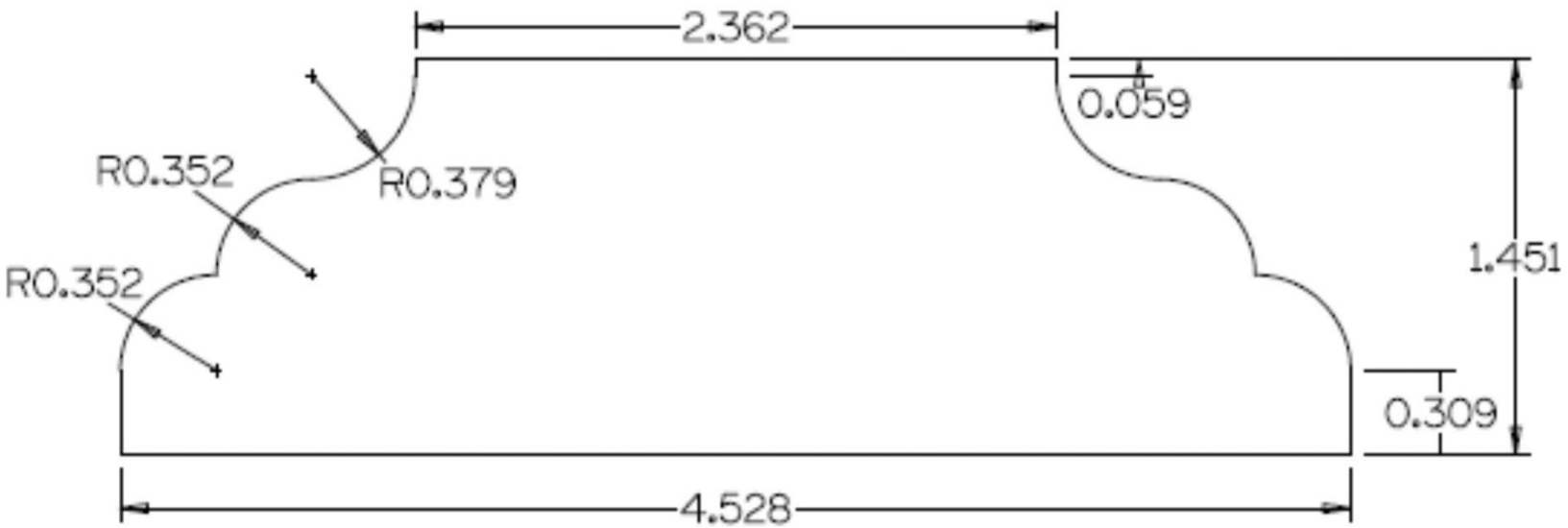
Metal

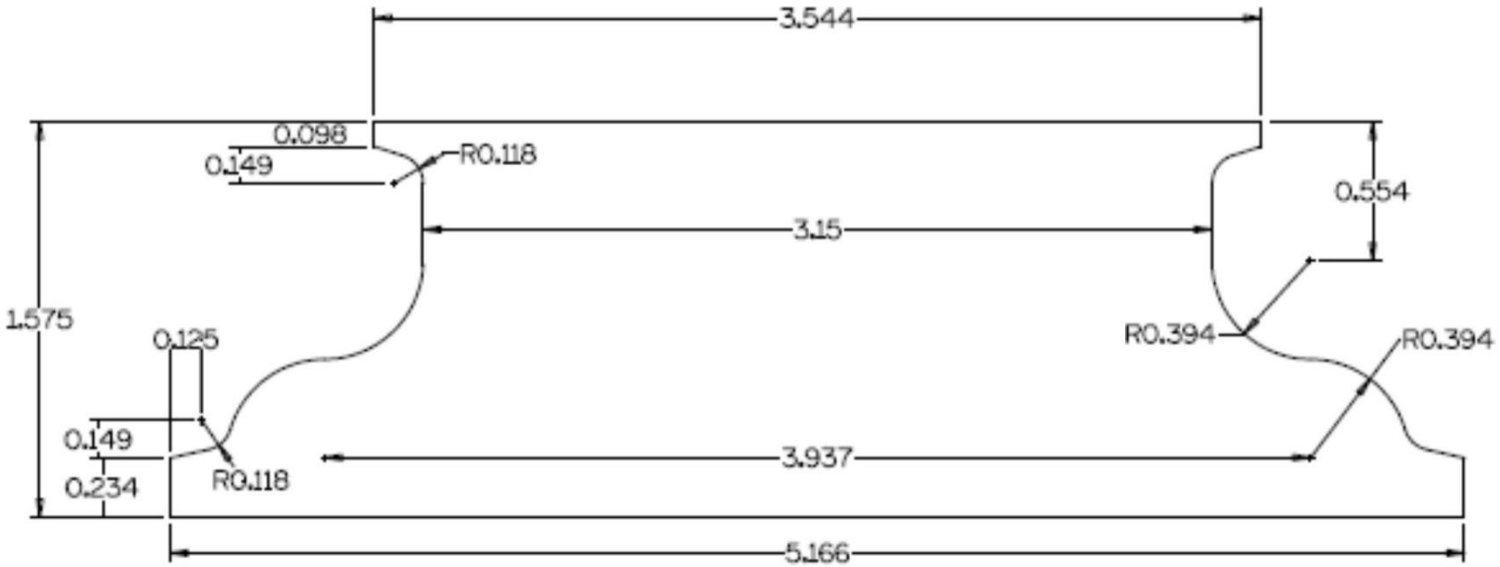


Polisher



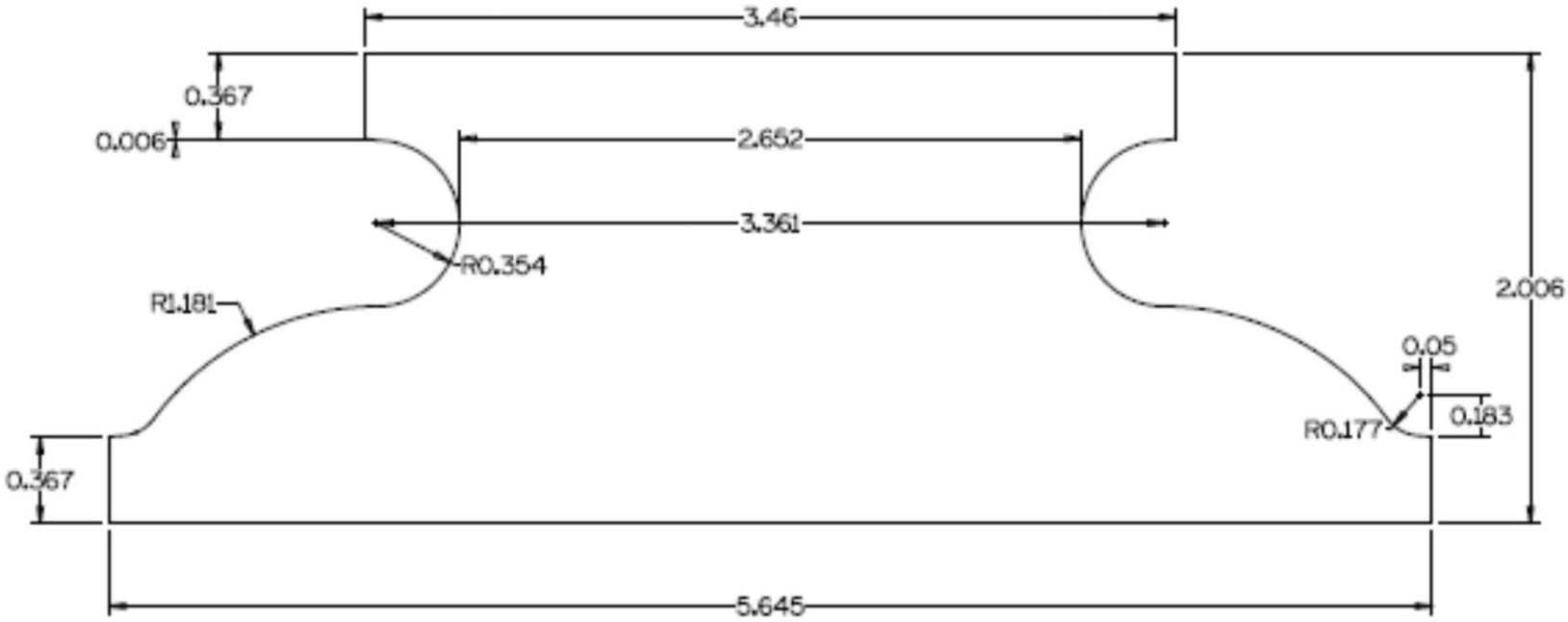
Profile Q-30
Metal and Polisher





Profile FS-30mm R3

id=80mm od=131.2mm



Profile FG-33mm

id=68mm od=120mm



Profile T-30mm R3

id=52mm od=66mm

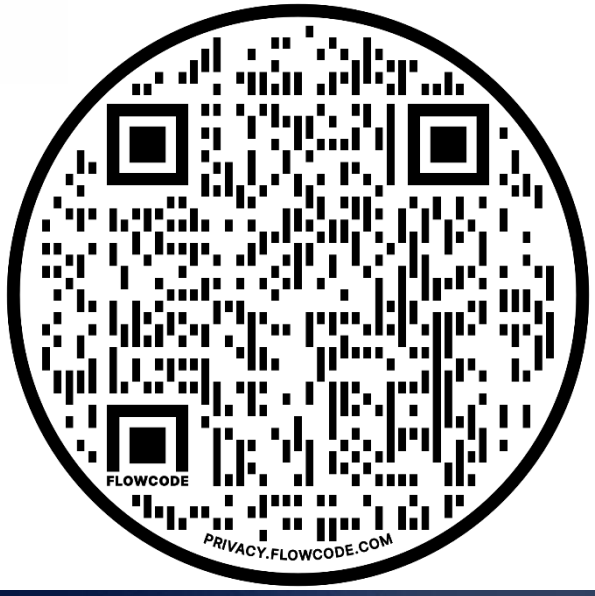
Section 4

Building the Tool Library for Titan

Videos for this section can be found on our website's service and training guides.
https://www.parkindustries.com/stone/service-support/guides/

Suggested videos/guides include

CREATING TOOLS IN ALPHACAM





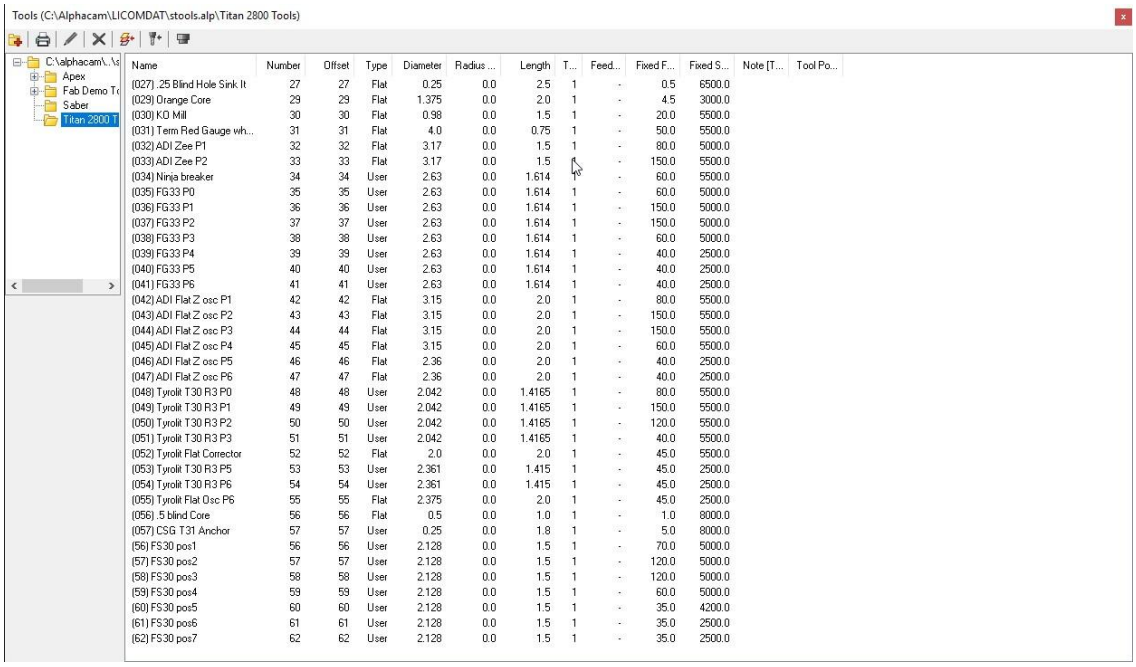
Building the Tool Library for the Titan

Tools must be entered and all parameters setup in the Tool library before they can be used to create cuts.

User Defined tools must be properly defined (drawn) before they can be entered into the Tool Library. The needed information to draw these tools correctly is in tooling portion of the CAD examples (Section 1).

The tools are saved (by default) in C:\Alphacam\LICOMDAT\stools.alp

Choose the **Machine** tab and then pick
Select Tool to open and view the **Tool Library**.



Titan Tool Library Vs. AlphaCam Tool Library

A library of tools and tool information is stored in both the AlphaCam software and the Titan machine.

When a program is created in AlphaCam, a portion of the information on how the tools will run, comes from the AlphaCam library, and a portion of the information comes from the Titan machine library.

For Example:

The **RPMs** and **Feed Rate** of a tool will come from the AlphaCam library.

The **Length** and **Diameter** of a tool will come from the Titan machine library.

In Addition, there is some information that can be set in either the AlphaCam library, or the Titan machine library.

For Example:

If we needed to raise the height of a core bit, we could either adjust the "**Default Z depth**" setting in the Titan machine library. Or adjust the "**Final Depth**" setting in the AlphaCam software.

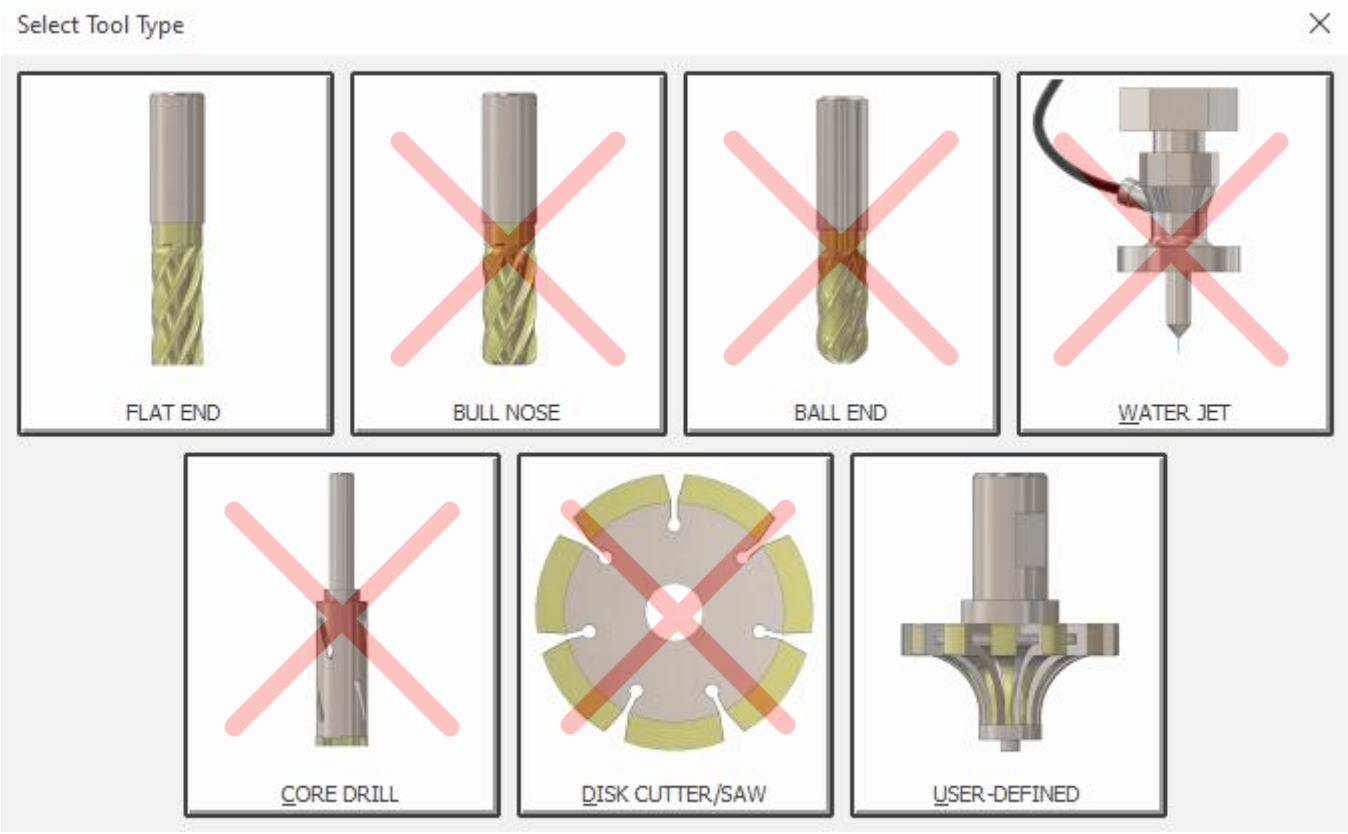
Or:

If we needed to move the mill bit/finger bit away from the finished edge to leave mate- rial behind for the following tools, we could either adjust the "**Removal**" setting in the Titan machine library. Or adjust the "**Stock To Be Left**" setting in the AlphaCam soft- ware.

NOTE: If a value is entered at the Titan machine library and the AlphaCam soft- ware in either example, the two values will be added together. So be sure to make the adjustment in only one location and the other location should be set to zero.

Define Tool

To build or add tools to the Tool Library, you will define the tool by “Selecting Tool Type”. This can be done by going to the **Machine** tab and choosing “**Define Tool**”. In the “Select Tool Type window choose either Flat End or User Defined.

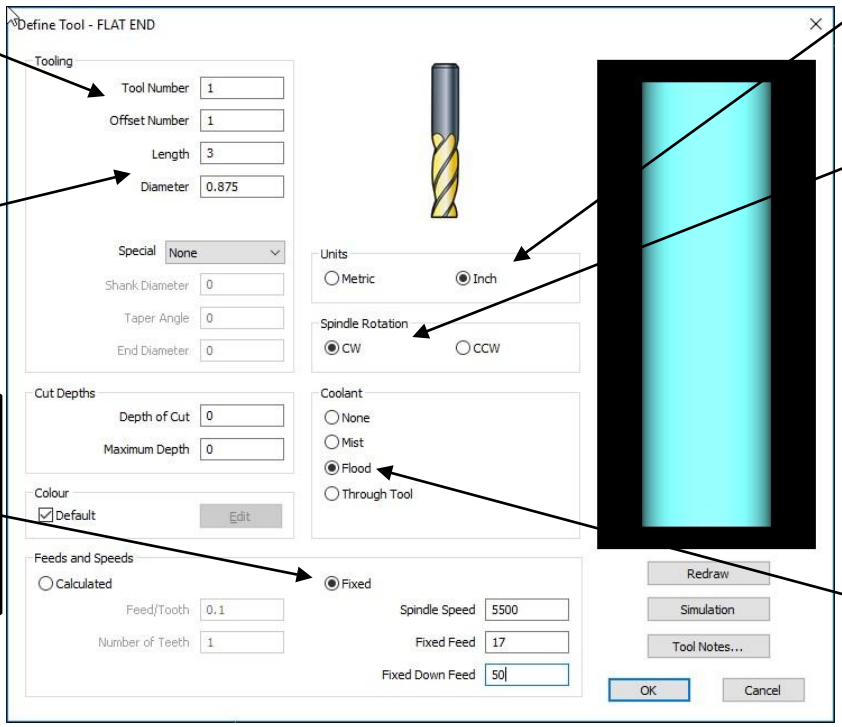


Flat End Tool

Tool Number and Offset Number match the tool position at the machine

Length and Diameter should be set to the length and diameter of the tool. Length is the length of the cutting edge.

Fixed is always selected under feeds and speeds.
Spindle Speed = Tool RPM
Fixed Feed = Tool Feedrate
Fixed Down Feed = 50 for Edging.
When Coring/Drilling Fixed Feed and Fixed Down Feed should be the same value



Units always = Inch

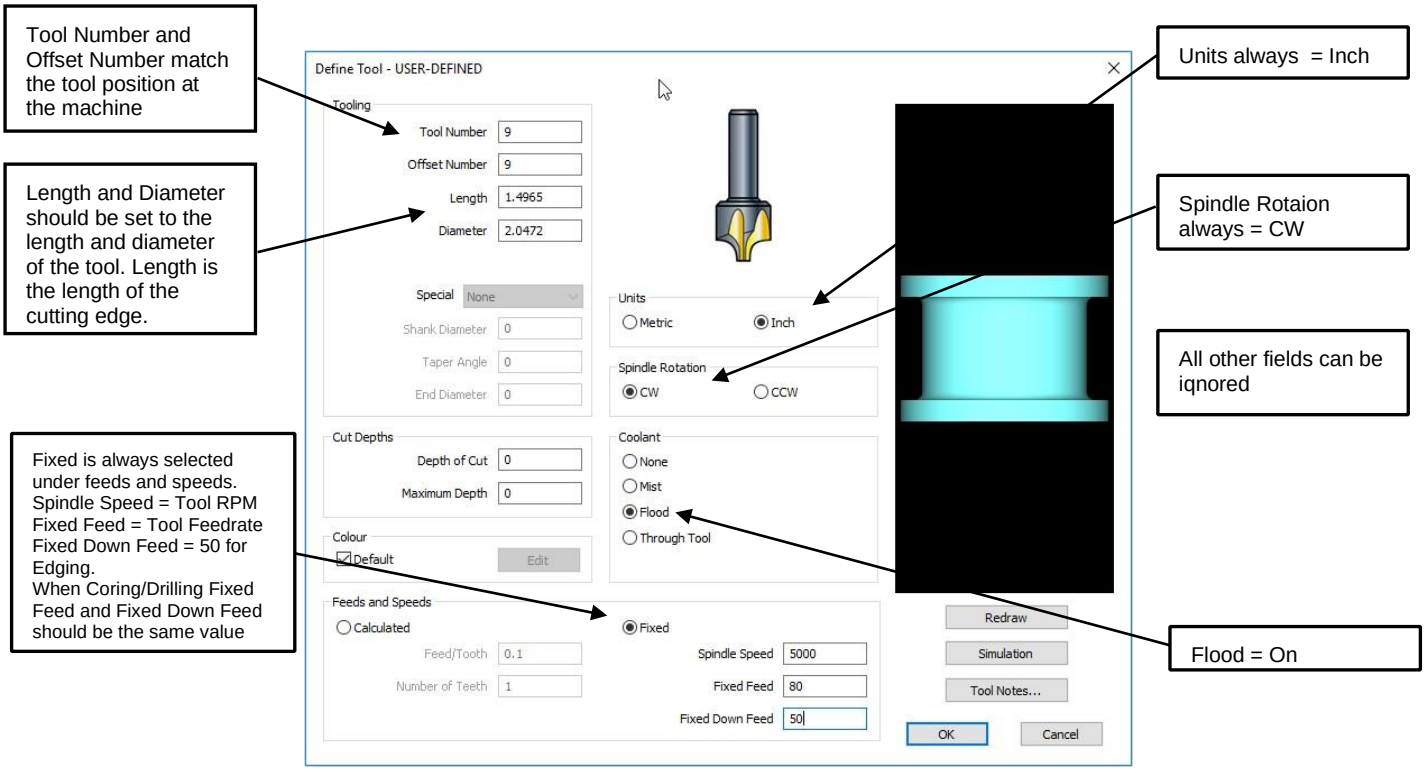
Spindle Rotation always = CW

All other fields can be ignored

Flood = On

User Defined Tool

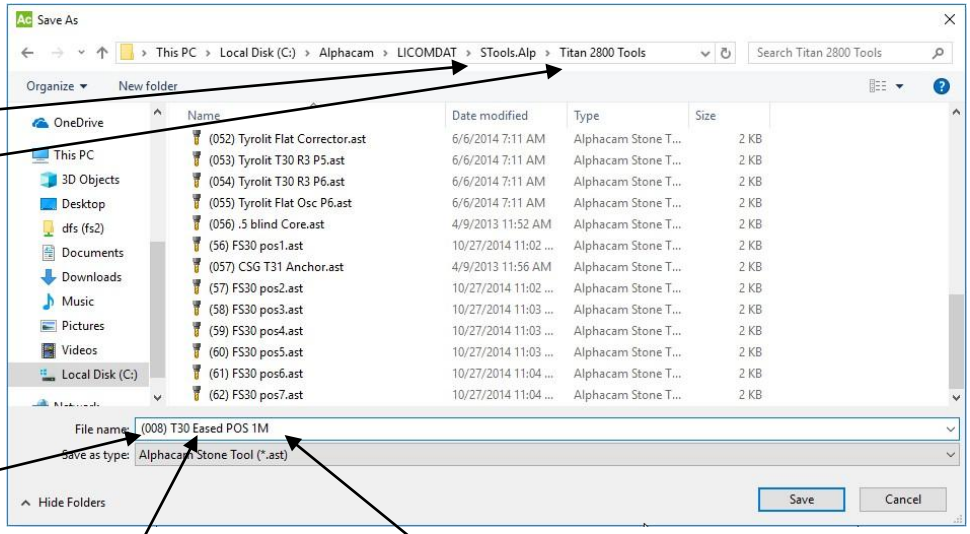
- When selecting user defined tool you must have a tool drawing open
- You will be prompted to pick the geometry defining the cross section, click the tool outline
- Verify the box appears acknowledging the line and circle were found, if not refer back to the tool drawing to find the problem
- Refer to the “Three Rules” of drawing a user defined tool



Tool Naming

- Use a tool name that you understand
- It is suggested to start with a three-digit number in parenthesis
- Use POS for the tool number within a profile set

Tools must be saved in **Licomdat** – **STOOLS.ALP** Additional folders may be created in **STOOLS.ALP** for different tool libraries



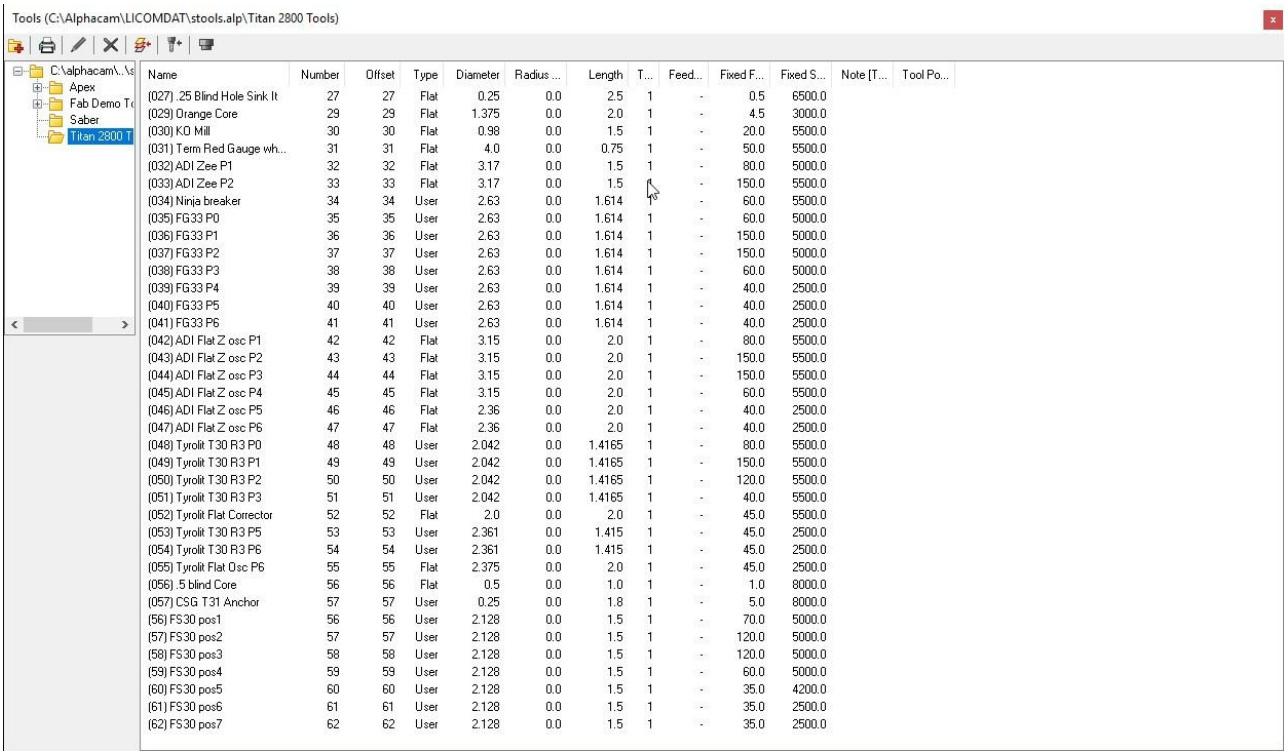
3 digit tool number in parenthesis that matches the tool number in the tool library on the machine .

Tool name

Tool position within the profile set

Tool Library Review

- Choose **Select Tool** from the **Machine** tab in AlphaCAM
- This window will display all of the Tool Parameters for each tool
- You may right click on a tool name and choose **Edit** to fix any errors



Name	Number	Offset	Type	Diameter	Radius ...	Length	T...	Feed...	Fixed F...	Fixed S...	Note [T...	Tool Po...
(027) .25 Blind Hole Sink It	27	27	Flat	0.25	0.0	2.5	1	-	0.5	6500.0		
(029) Orange Core	29	29	Flat	1.375	0.0	2.0	1	-	4.5	3000.0		
(030) KO Mill	30	30	Flat	0.98	0.0	1.5	1	-	20.0	5500.0		
(031) Term Red Gauge wh...	31	31	Flat	4.0	0.0	0.75	1	-	50.0	5500.0		
(032) ADI Zee P1	32	32	Flat	3.17	0.0	1.5	1	-	80.0	5000.0		
(033) ADI Zee P2	33	33	Flat	3.17	0.0	1.5	1	-	150.0	5500.0		
(034) Ninja breaker	34	34	User	2.63	0.0	1.614	1	-	60.0	5500.0		
(035) FG33 P0	35	35	User	2.63	0.0	1.614	1	-	60.0	5000.0		
(036) FG33 P1	36	36	User	2.63	0.0	1.614	1	-	150.0	5000.0		
(037) FG33 P2	37	37	User	2.63	0.0	1.614	1	-	150.0	5000.0		
(038) FG33 P3	38	38	User	2.63	0.0	1.614	1	-	60.0	5000.0		
(039) FG33 P4	39	39	User	2.63	0.0	1.614	1	-	40.0	2500.0		
(040) FG33 P5	40	40	User	2.63	0.0	1.614	1	-	40.0	2500.0		
(041) FG33 P6	41	41	User	2.63	0.0	1.614	1	-	40.0	2500.0		
(042) ADI Flat Z osc P1	42	42	Flat	3.15	0.0	2.0	1	-	80.0	5500.0		
(043) ADI Flat Z osc P2	43	43	Flat	3.15	0.0	2.0	1	-	150.0	5500.0		
(044) ADI Flat Z osc P3	44	44	Flat	3.15	0.0	2.0	1	-	150.0	5500.0		
(045) ADI Flat Z osc P4	45	45	Flat	3.15	0.0	2.0	1	-	60.0	5500.0		
(046) ADI Flat Z osc P5	46	46	Flat	2.36	0.0	2.0	1	-	40.0	2500.0		
(047) ADI Flat Z osc P6	47	47	Flat	2.36	0.0	2.0	1	-	40.0	2500.0		
(048) Tyrolit T30 R3 P0	48	48	User	2.042	0.0	1.4165	1	-	80.0	5500.0		
(049) Tyrolit T30 R3 P1	49	49	User	2.042	0.0	1.4165	1	-	150.0	5500.0		
(050) Tyrolit T30 R3 P2	50	50	User	2.042	0.0	1.4165	1	-	120.0	5500.0		
(051) Tyrolit T30 R3 P3	51	51	User	2.042	0.0	1.4165	1	-	40.0	5500.0		
(052) Tyrolit Flat Corrector	52	52	Flat	2.0	0.0	2.0	1	-	45.0	5500.0		
(053) Tyrolit T30 R3 P5	53	53	User	2.361	0.0	1.415	1	-	45.0	2500.0		
(054) Tyrolit T30 R3 P6	54	54	User	2.361	0.0	1.415	1	-	45.0	2500.0		
(055) Tyrolit Flat Osc P6	55	55	Flat	2.375	0.0	2.0	1	-	45.0	2500.0		
(056) .5 blind Core	56	56	Flat	0.5	0.0	1.0	1	-	1.0	8000.0		
(057) CSG T31 Anchor	57	57	User	0.25	0.0	1.8	1	-	5.0	8000.0		
(56) FS30 pos1	56	56	User	2.128	0.0	1.5	1	-	70.0	5000.0		
(57) FS30 pos2	57	57	User	2.128	0.0	1.5	1	-	120.0	5000.0		
(58) FS30 pos3	58	58	User	2.128	0.0	1.5	1	-	120.0	5000.0		
(59) FS30 pos4	59	59	User	2.128	0.0	1.5	1	-	60.0	5000.0		
(60) FS30 pos5	60	60	User	2.128	0.0	1.5	1	-	35.0	4200.0		
(61) FS30 pos6	61	61	User	2.128	0.0	1.5	1	-	35.0	2500.0		
(62) FS30 pos7	62	62	User	2.128	0.0	1.5	1	-	35.0	2500.0		

Notes:

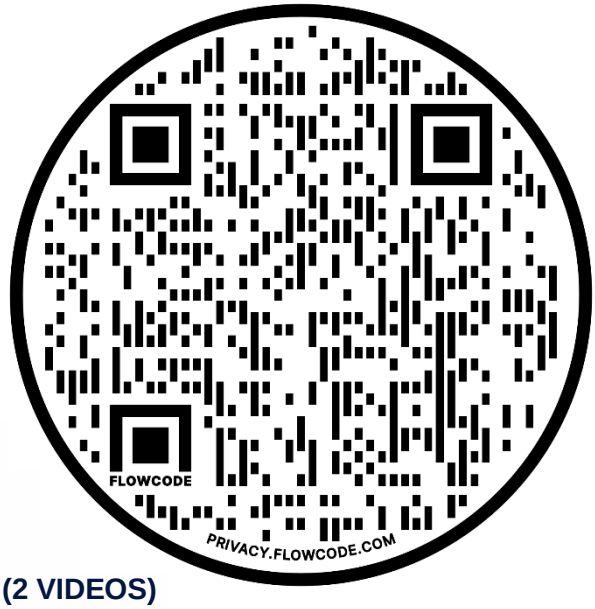
Section 5

Creating Tool Paths with “Cut Shape”

Videos for this section can be found on our website’s service and training guides.
https://www.parkindustries.com/stone/service-support/guides/

Suggested videos/guides include

USING CUT SHAPE – BASICS (VIDEO+ LINKS)
USING CUT HOLES (VIDEO + LINKS)
BASIC TITAN PROGRAMMING (2 VIDEOS+LINKS)
USING ADVANCED TOOL DIRECTION METHODS(VIDEO)
EDITING LEAD IN & OUTS (VIDEO)
REDUCING TOOL CHANGES ON THE TITAN (VIDEO)
ADDING A VACUUM POD TO THE TITAN TABLE TEMPLATE (2 VIDEOS)



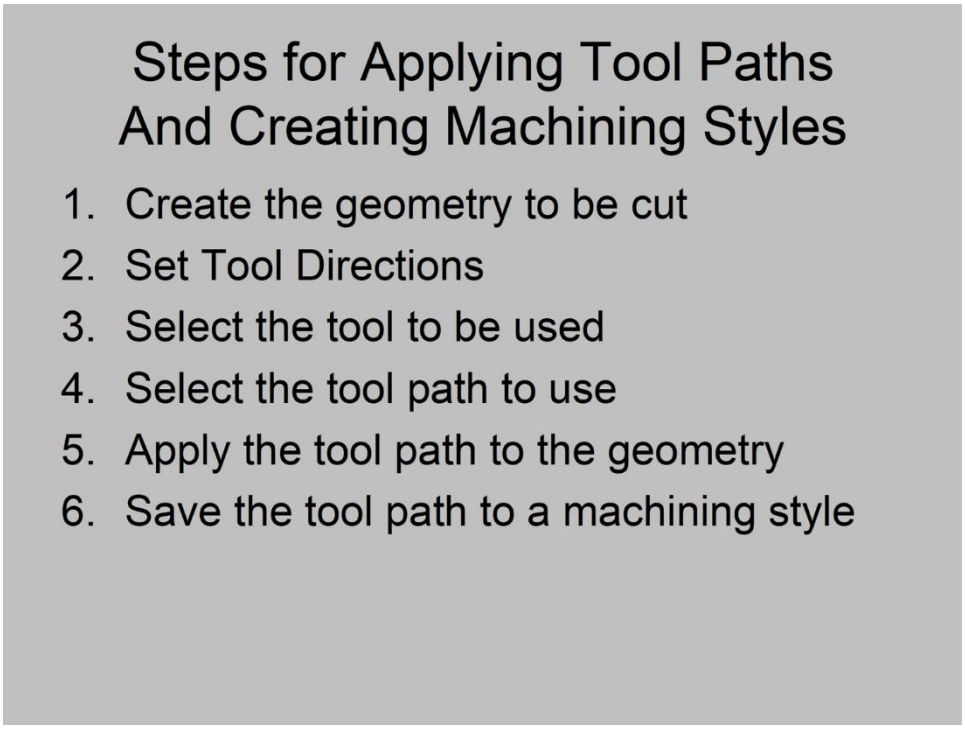
SERVICE & TRAINING GUIDES

Creating Tool Paths

Tool paths are applied to geometry to create part programs.

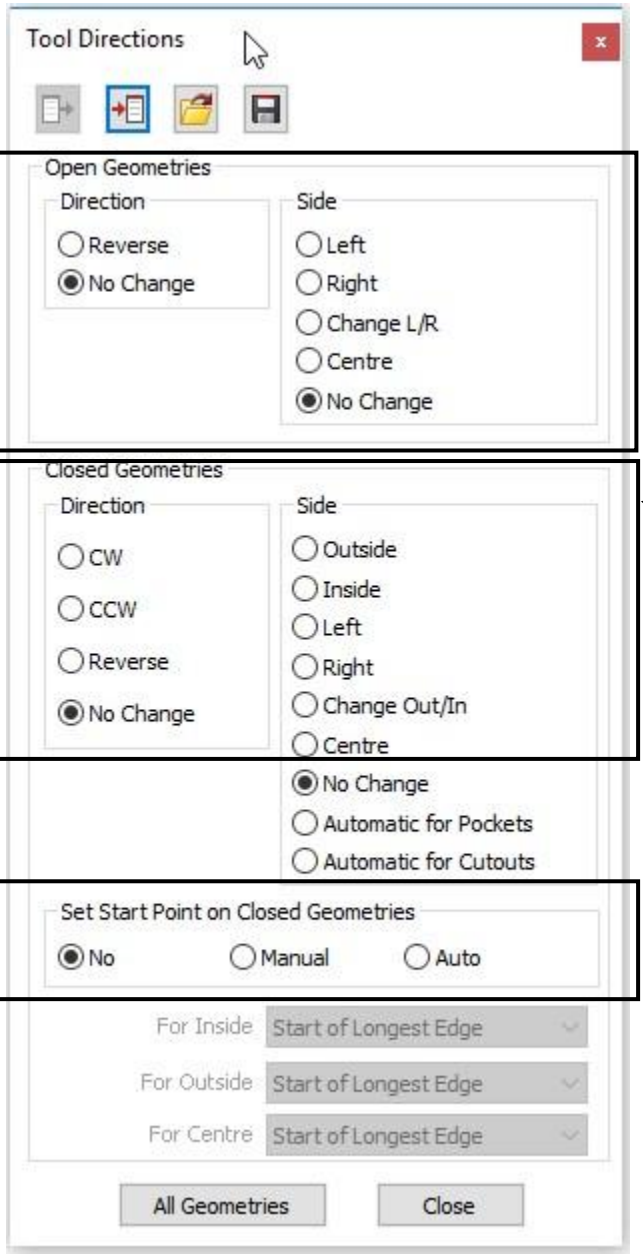
The order that the tool paths are applied will be the default sequence of operations for the program.

The Tool Paths may be saved as “Styles” after they are applied.



Tool Directions

Open the “Tool Direction” window by choosing it from the Machine tab. It will only be available if there is geometry in the drawing area. Leave the window open while setting directions on your parts. When you close the window, the tool directions will stay applied.

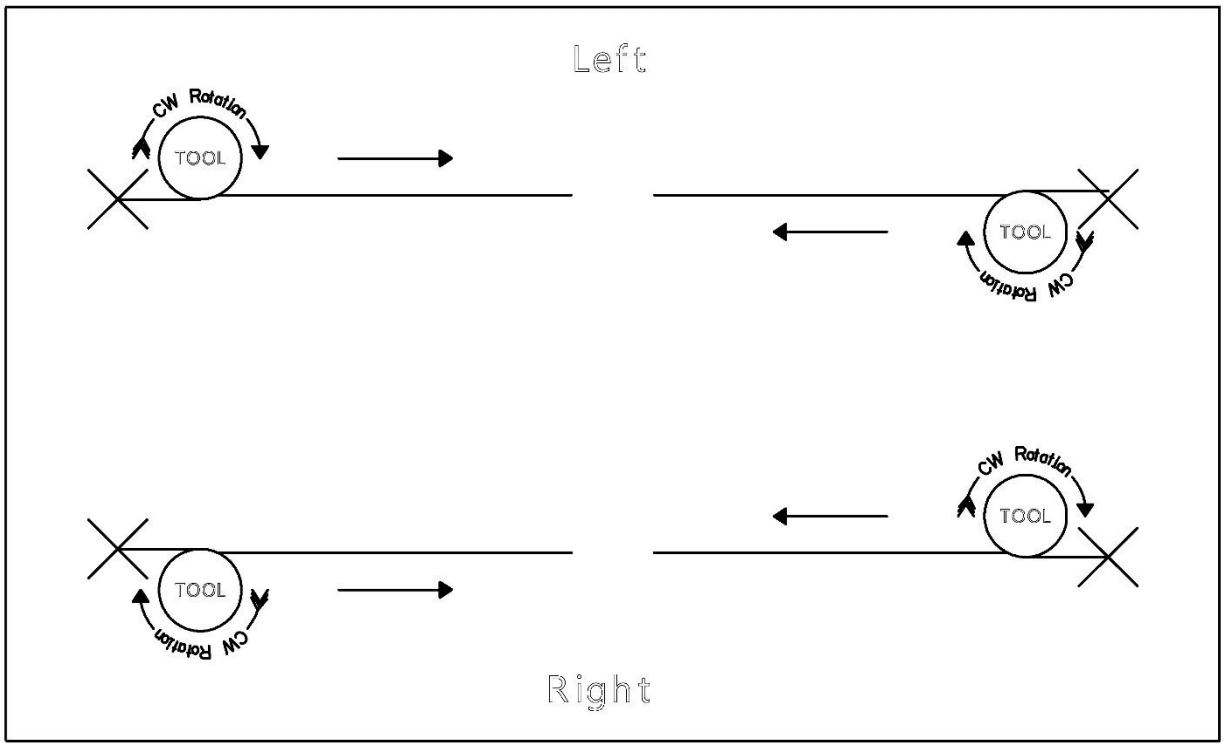


This section is used only for **Open Geometries**. Most of the time it should be set to **Reverse** and **Left**.

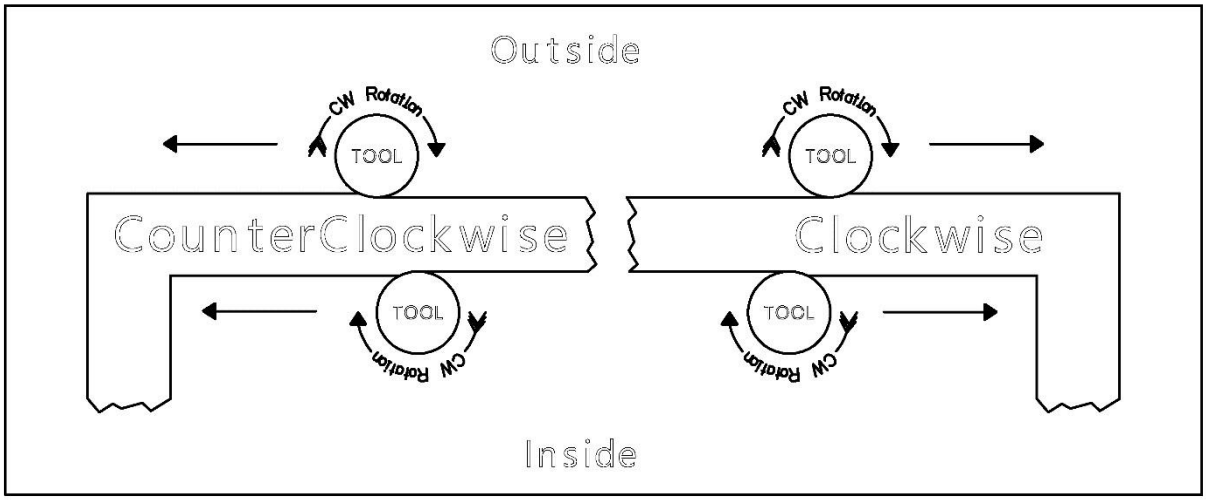
This section is used only for **Closed Geometries**. Most of the time it should be set to **CW and Outside** or **CCW for Inside.**

When applying Tool Directions to **Open Shapes**, Choose **No**, and **Manual** for **Closed shapes**.

Open Geometries

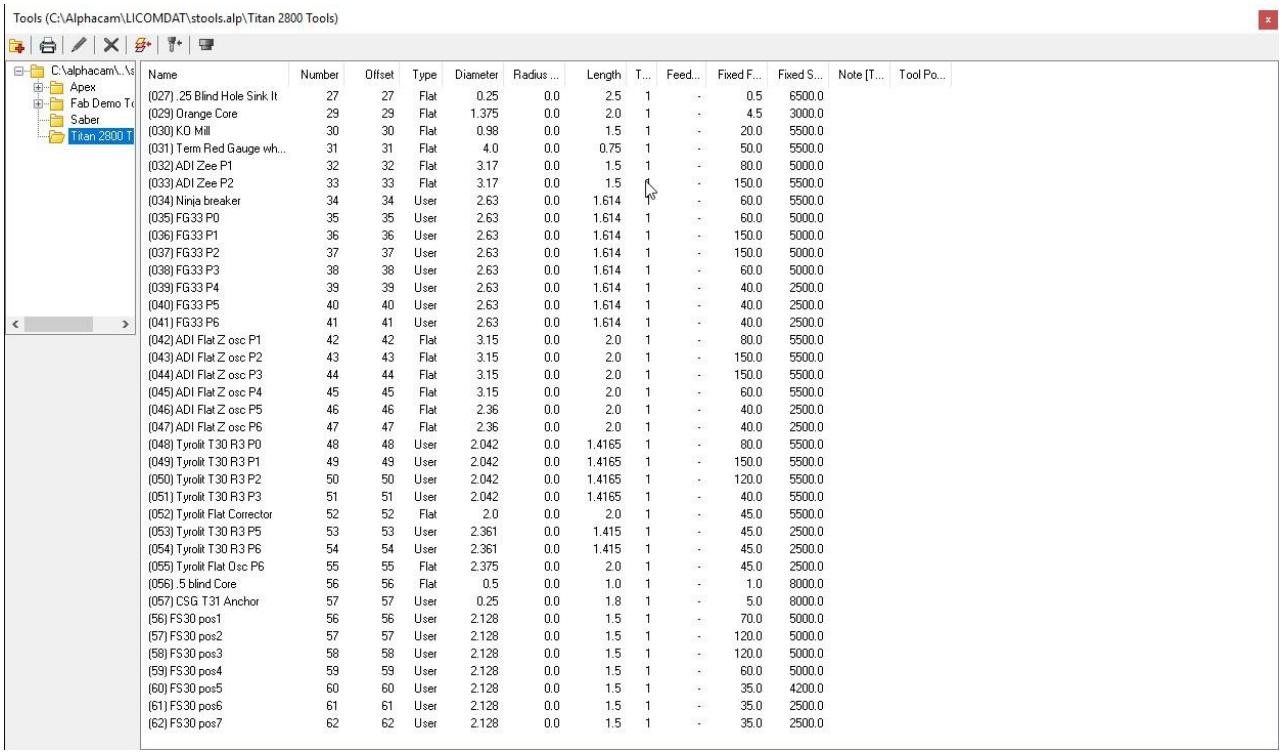


Closed Geometries



Selecting a tool

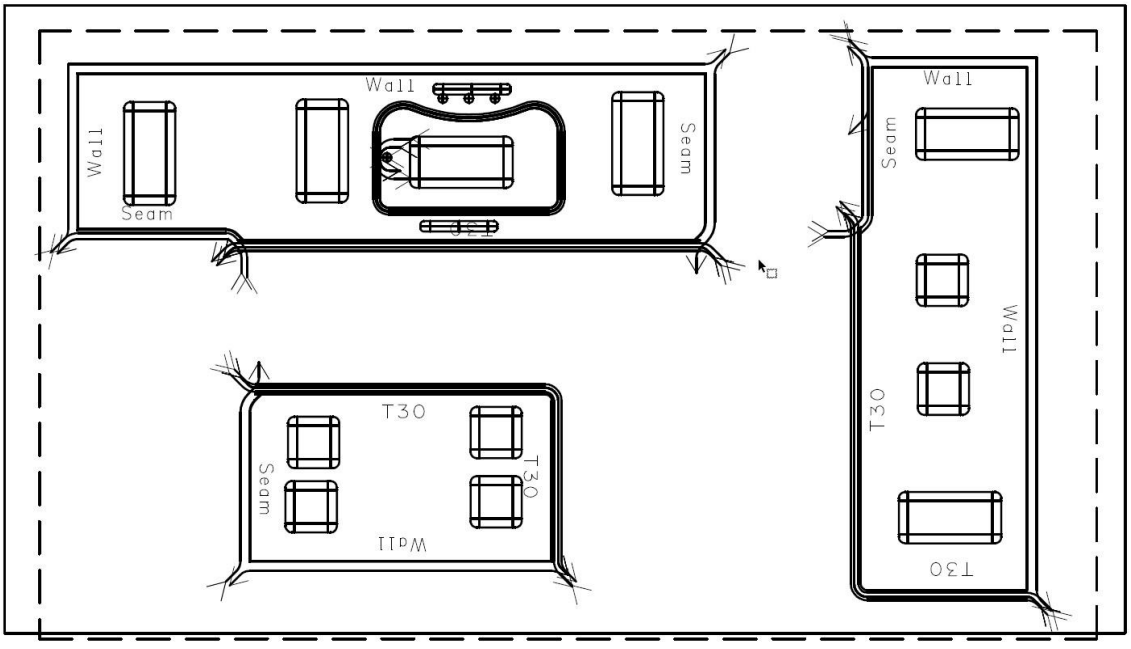
Before Cutting Shapes or Cutting Holes the proper tool must be selected. The current selected tool will remain active until a new tool is selected. To select a tool, choose **Select Tool** from the **Machine Tab.** Then double click on the name of the tool that you would like to use and click anywhere in the drawing area to use/activate it.



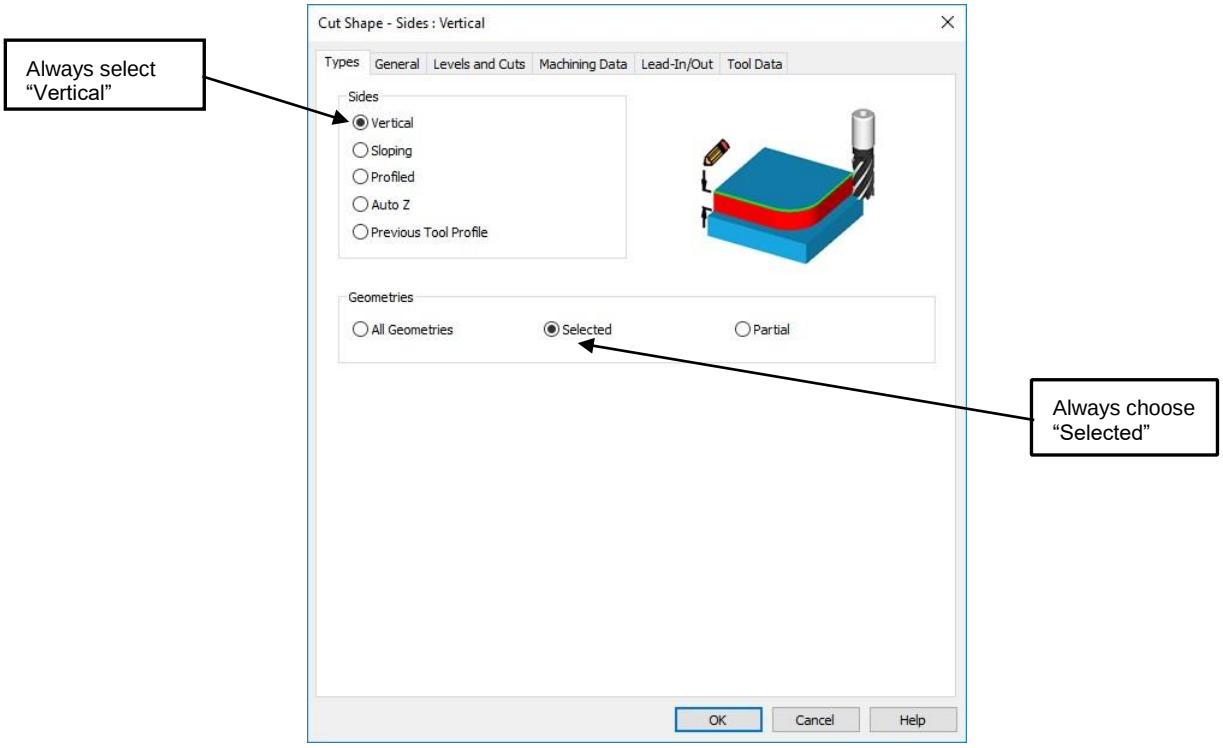
Name	Number	Offset	Type	Diameter	Radius ...	Length	T...	Feed...	Fixed F...	Fixed S...	Note [T...	Tool Po...
(027) .25 Blind Hole Sink It	27	27	Flat	0.25	0.0	2.5	1	-	0.5	6500.0		
(029) Orange Core	29	29	Flat	1.375	0.0	2.0	1	-	4.5	3000.0		
(030) KO Mill	30	30	Flat	0.98	0.0	1.5	1	-	20.0	5500.0		
(031) Term Red Gauge wh...	31	31	Flat	4.0	0.0	0.75	1	-	50.0	5500.0		
(032) ADI Zee P1	32	32	Flat	3.17	0.0	1.5	1	-	80.0	5000.0		
(033) ADI Zee P2	33	33	Flat	3.17	0.0	1.5	1	-	150.0	5500.0		
(034) Ninja breaker	34	34	User	2.63	0.0	1.614	1	-	60.0	5500.0		
(035) FG33 P0	35	35	User	2.63	0.0	1.614	1	-	60.0	5000.0		
(036) FG33 P1	36	36	User	2.63	0.0	1.614	1	-	150.0	5000.0		
(037) FG33 P2	37	37	User	2.63	0.0	1.614	1	-	150.0	5000.0		
(038) FG33 P3	38	38	User	2.63	0.0	1.614	1	-	60.0	5000.0		
(039) FG33 P4	39	39	User	2.63	0.0	1.614	1	-	40.0	2500.0		
(040) FG33 P5	40	40	User	2.63	0.0	1.614	1	-	40.0	2500.0		
(041) FG33 P6	41	41	User	2.63	0.0	1.614	1	-	40.0	2500.0		
(042) ADI Flat Z osc P1	42	42	Flat	3.15	0.0	2.0	1	-	80.0	5500.0		
(043) ADI Flat Z osc P2	43	43	Flat	3.15	0.0	2.0	1	-	150.0	5500.0		
(044) ADI Flat Z osc P3	44	44	Flat	3.15	0.0	2.0	1	-	150.0	5500.0		
(045) ADI Flat Z osc P4	45	45	Flat	3.15	0.0	2.0	1	-	60.0	5500.0		
(046) ADI Flat Z osc P5	46	46	Flat	2.36	0.0	2.0	1	-	40.0	2500.0		
(047) ADI Flat Z osc P6	47	47	Flat	2.36	0.0	2.0	1	-	40.0	2500.0		
(048) Tyrolit T30 R3 P0	48	48	User	2.042	0.0	1.4165	1	-	80.0	5500.0		
(049) Tyrolit T30 R3 P1	49	49	User	2.042	0.0	1.4165	1	-	150.0	5500.0		
(050) Tyrolit T30 R3 P2	50	50	User	2.042	0.0	1.4165	1	-	120.0	5500.0		
(051) Tyrolit T30 R3 P3	51	51	User	2.042	0.0	1.4165	1	-	40.0	5500.0		
(052) Tyrolit Flat Corrector	52	52	Flat	2.0	0.0	2.0	1	-	45.0	5500.0		
(053) Tyrolit T30 R3 P5	53	53	User	2.361	0.0	1.415	1	-	45.0	2500.0		
(054) Tyrolit T30 R3 P6	54	54	User	2.361	0.0	1.415	1	-	45.0	2500.0		
(055) Tyrolit Flat Osc P6	55	55	Flat	2.375	0.0	2.0	1	-	45.0	2500.0		
(056) .5 blind Core	56	56	Flat	0.5	0.0	1.0	1	-	1.0	8000.0		
(057) CSG T31 Anchor	57	57	User	0.25	0.0	1.8	1	-	5.0	8000.0		
(56) FS30 pos1	56	56	User	2.128	0.0	1.5	1	-	70.0	5000.0		
(57) FS30 pos2	57	57	User	2.128	0.0	1.5	1	-	120.0	5000.0		
(58) FS30 pos3	58	58	User	2.128	0.0	1.5	1	-	120.0	5000.0		
(59) FS30 pos4	59	59	User	2.128	0.0	1.5	1	-	60.0	5000.0		
(60) FS30 pos5	60	60	User	2.128	0.0	1.5	1	-	35.0	4200.0		
(61) FS30 pos6	61	61	User	2.128	0.0	1.5	1	-	35.0	2500.0		
(62) FS30 pos7	62	62	User	2.128	0.0	1.5	1	-	35.0	2500.0		

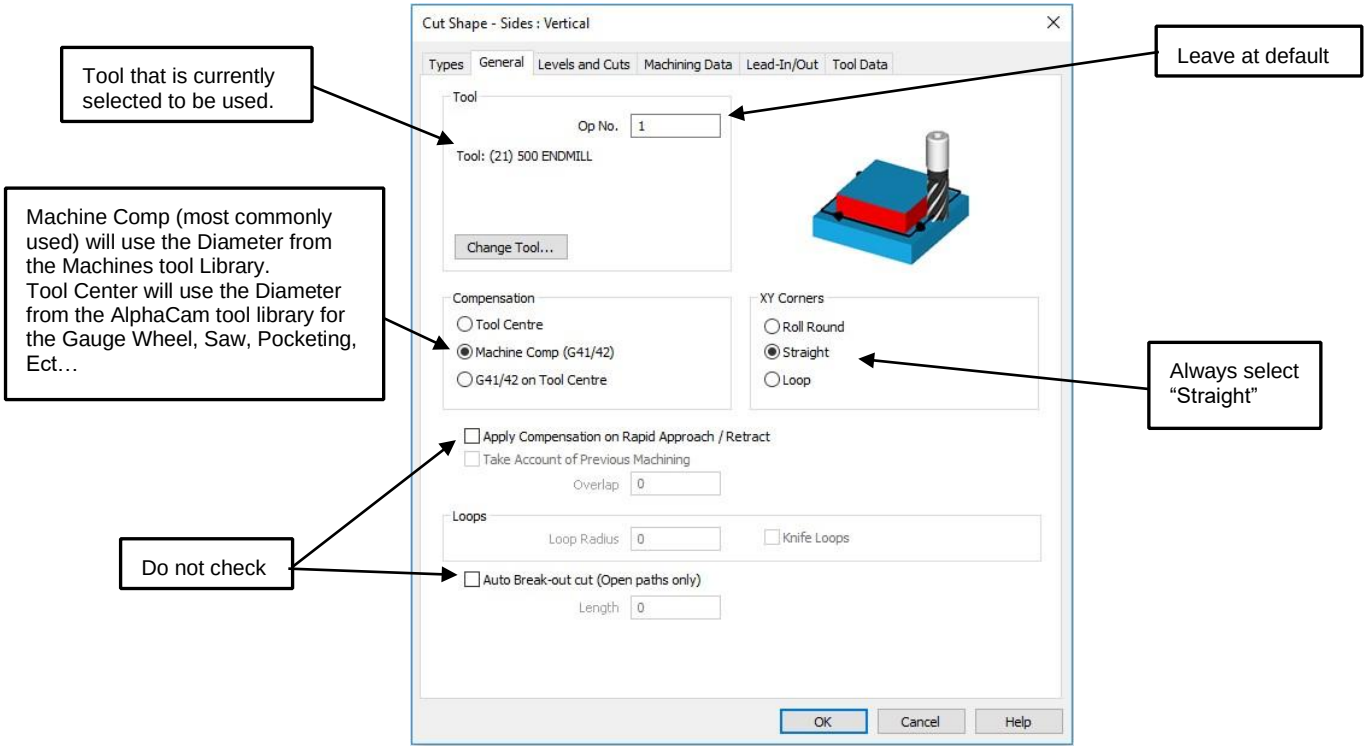
Applying cuts

To apply a cut to Geometry, choose either **Cut Shape** or **Cut Hole** from the **Machine Tab.** These icons will only be available if you have geometry in the drawing area and a tool selected. Tool Direction should also be set on all geometries to be cut.



Cut Shape


Always select "Vertical"
Cut Shape - Sides : Vertical
Types
General
Levels and Cuts
Machining Data
Lead-In/Out
Tool Data
Sides
Vertical
Sloping
Profiled
Auto Z
Previous Tool Profile
Geometries
All Geometries
Selected
Partial
OK
Cancel
Help
Always choose "Selected"


Tool that is currently selected to be used.
Cut Shape - Sides : Vertical
Types
General
Levels and Cuts
Machining Data
Lead-In/Out
Tool Data
Leave at default
Tool
Op No.
1
Tool: (21) 500 ENDMILL
Change Tool...
Machine Comp (most commonly used) will use the Diameter from the Machines tool Library.
Tool Center will use the Diameter from the AlphaCam tool library for the Gauge Wheel, Saw, Pocketing, Ect…
Compensation
Tool Centre
Machine Comp (G41/42)
G41/42 on Tool Centre
XY Corners
Roll Round
Straight
Loop
Always select "Straight"
Apply Compensation on Rapid Approach / Retract
Take Account of Previous Machining
Overlap
0
Loops
Loop Radius
0
Knife Loops
Do not check
Auto Break-out cut (Open paths only)
Length
0
OK
Cancel
Help

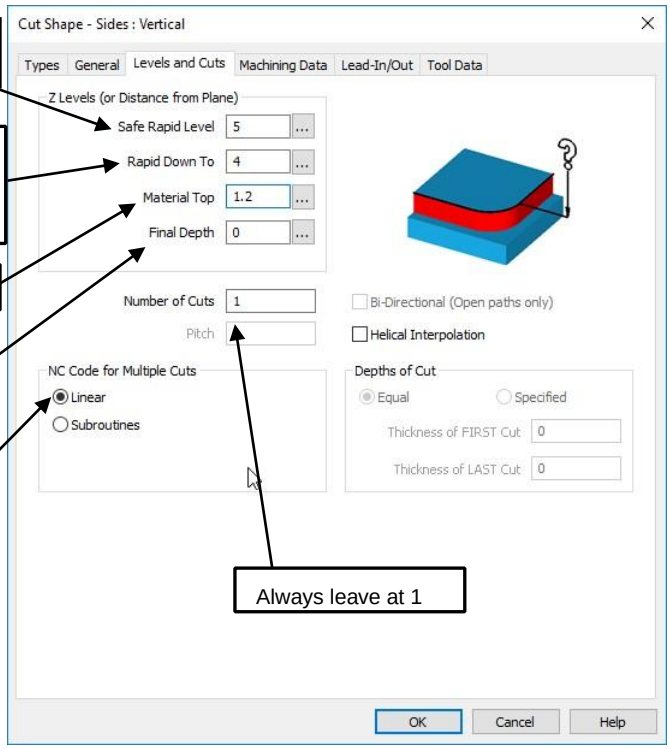
The Height the tool will rapid at in the X and Y direction

The Z Height the tool will rapid to in Z Typically set to 4, unless the tool is a Core Bit, Mill Bit, or Gauge Wheel… Then use 2.

The Height and thickness of material

The Z Height the tool will cut at, Usually zero unless Coring, Gauging, or Blading

Always Select “Linear”

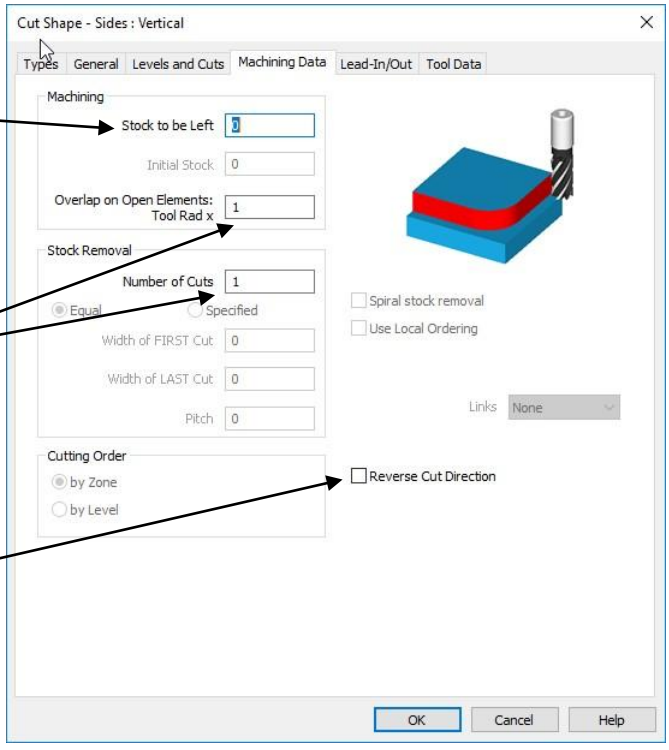


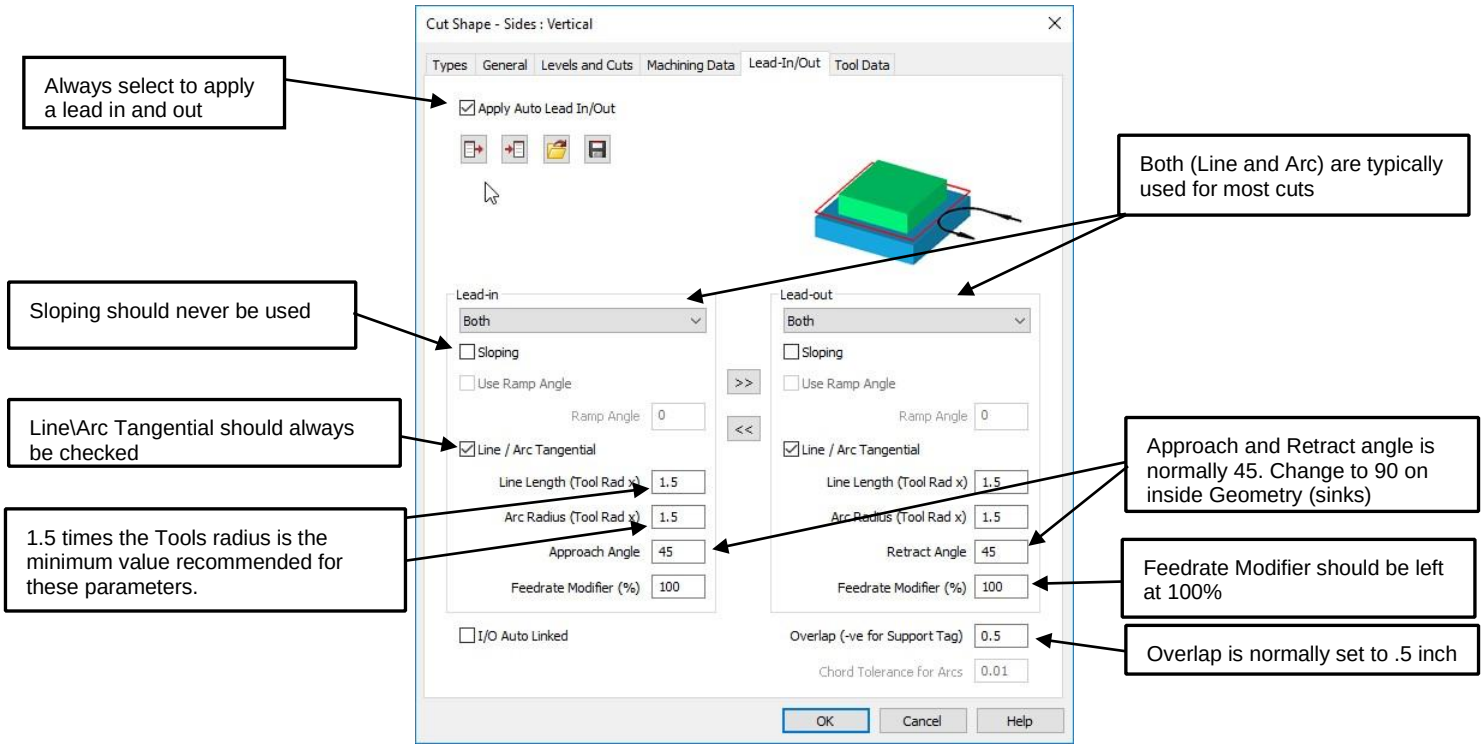
Always leave at 1

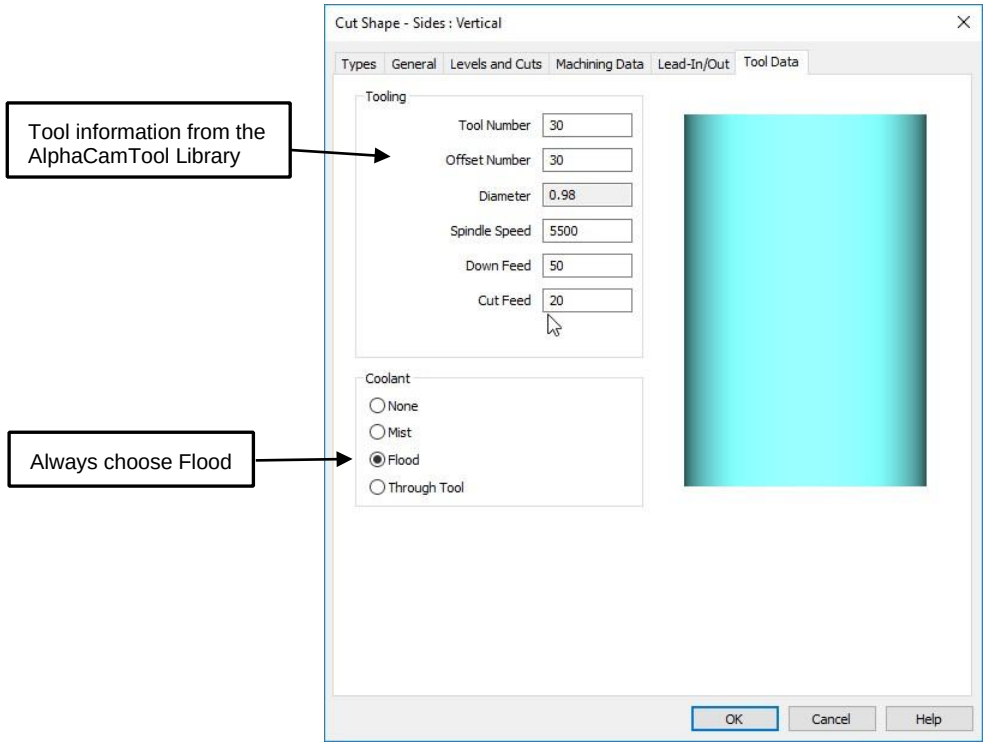
The amount of material the tool will leave behind. Typically zero unless running a pre-pass, Mill or Z wheel

Always 1

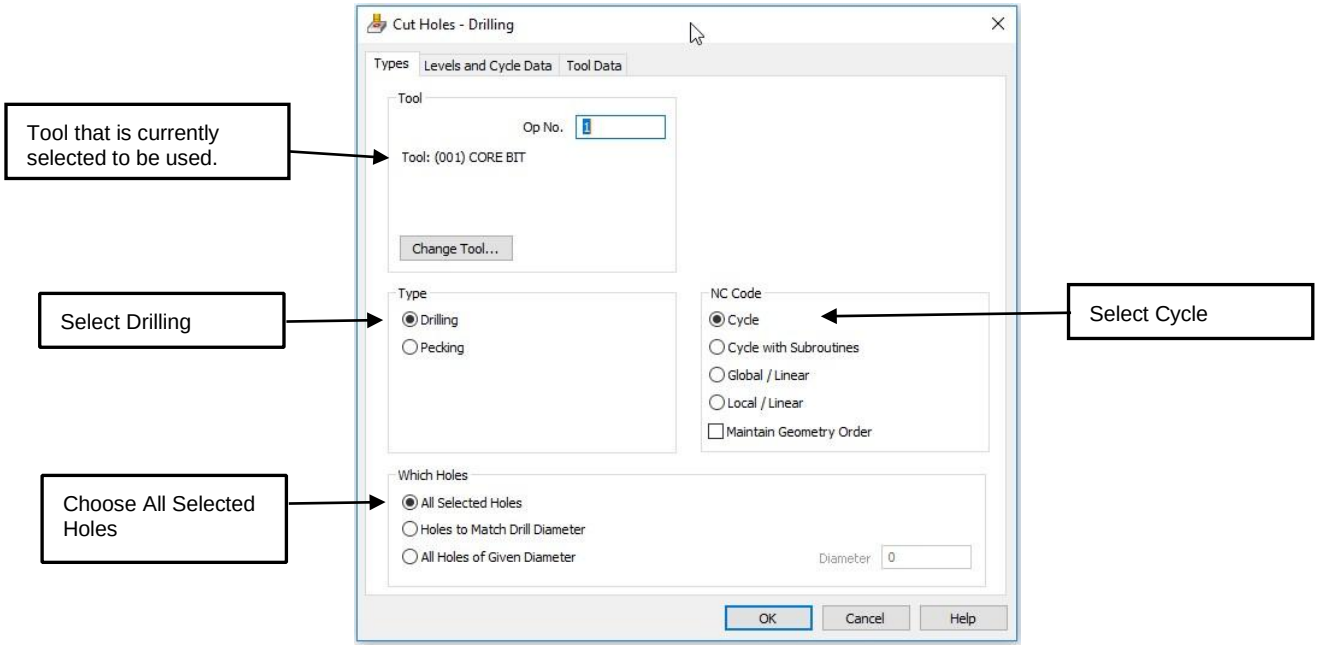
Do not check

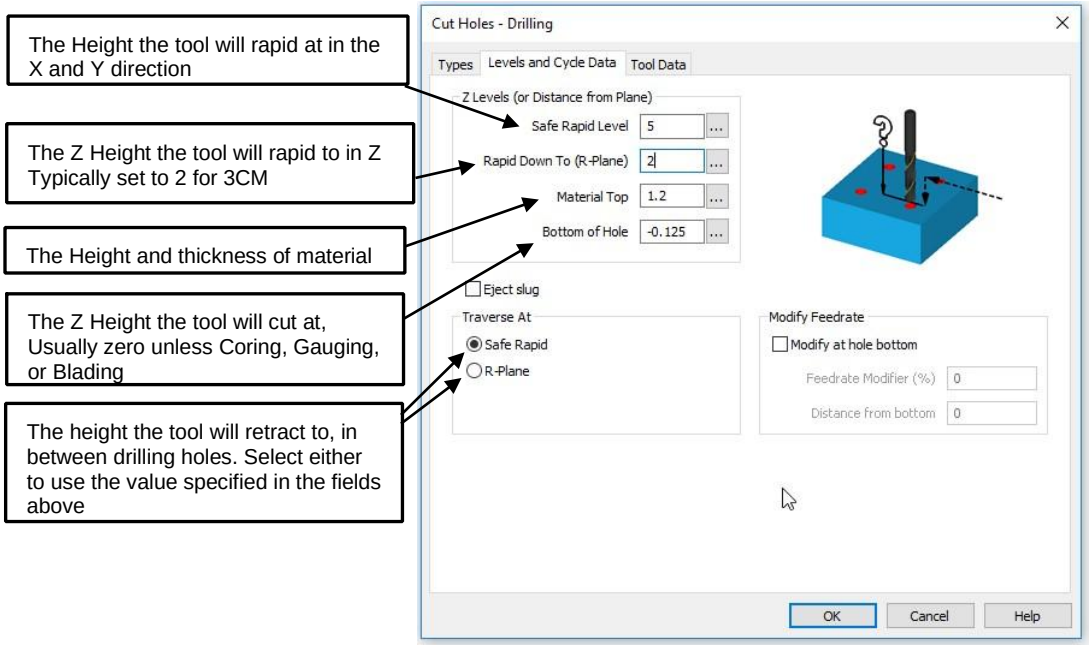


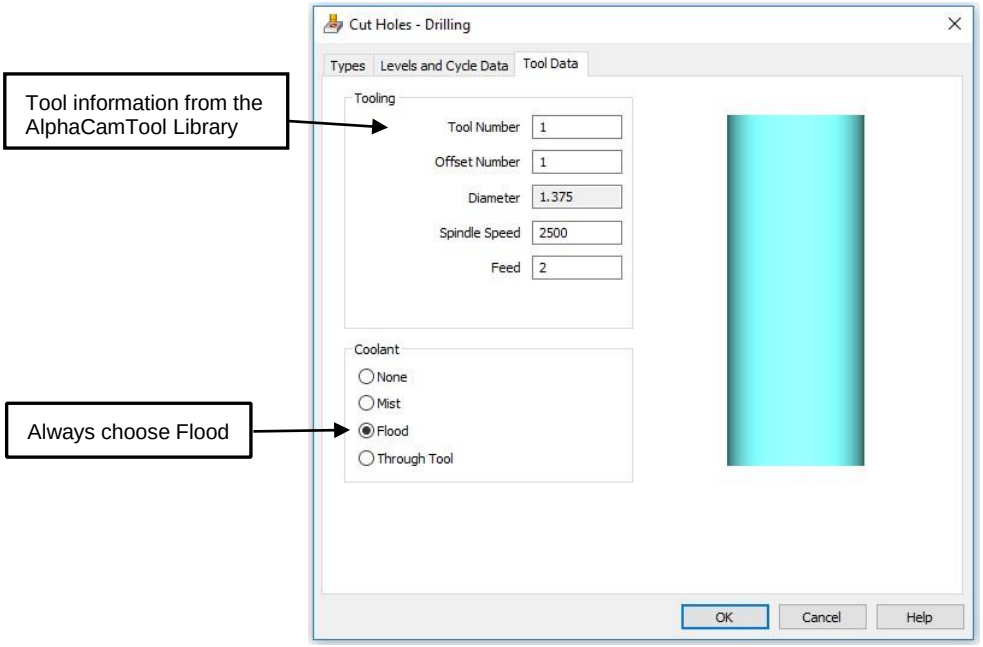

Always select to apply a lead in and out
Both (Line and Arc) are typically used for most cuts
Sloping should never be used
Line\Arc Tangential should always be checked
1.5 times the Tools radius is the minimum value recommended for these parameters.
Approach and Retract angle is normally 45. Change to 90 on inside Geometry (sinks)
Feedrate Modifier should be left at 100%
Overlap is normally set to .5 inch
Cut Shape - Sides : Vertical
Types
General
Levels and Cuts
Machining Data
Lead-In/Out
Tool Data
Apply Auto Lead In/Out
Lead-in
Lead-out
Both
Sloping
Use Ramp Angle
Ramp Angle
Line / Arc Tangential
Line Length (Tool Rad x)
Arc Radius (Tool Rad x)
Approach Angle
Retract Angle
Feedrate Modifier (%)
I/O Auto Linked
Overlap (-ve for Support Tag)
Chord Tolerance for Arcs
OK
Cancel
Help


Tool information from the AlphaCamTool Library
Always choose Flood
Cut Shape - Sides : Vertical
Types
General
Levels and Cuts
Machining Data
Lead-In/Out
Tool Data
Tooling
Tool Number
Offset Number
Diameter
Spindle Speed
Down Feed
Cut Feed
Coolant
None
Mist
Flood
Through Tool
OK
Cancel
Help

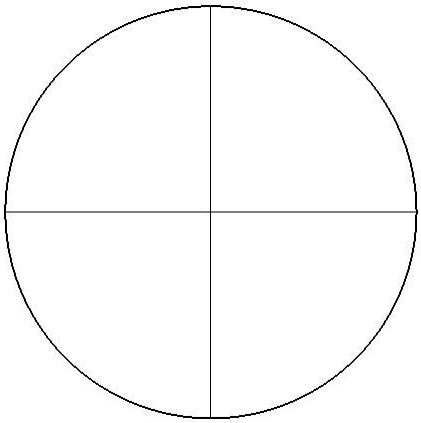
Cut Hole


Tool that is currently selected to be used.
Select Drilling
Choose All Selected Holes
Select Cycle
Cut Holes - Drilling
Types
Levels and Cycle Data
Tool Data
Tool
Op No.
1
Tool: (001) CORE BIT
Change Tool...
Type
Drilling
Pecking
NC Code
Cycle
Cycle with Subroutines
Global / Linear
Local / Linear
Maintain Geometry Order
Which Holes
All Selected Holes
Holes to Match Drill Diameter
All Holes of Given Diameter
Diameter
0
OK
Cancel
Help


The Height the tool will rapid at in the X and Y direction
The Z Height the tool will rapid to in Z Typically set to 2 for 3CM
The Height and thickness of material
The Z Height the tool will cut at, Usually zero unless Coring, Gauging, or Blading
The height the tool will retract to, in between drilling holes. Select either to use the value specified in the fields above
Cut Holes - Drilling
Types
Levels and Cycle Data
Tool Data
Z Levels (or Distance from Plane)
Safe Rapid Level
5
Rapid Down To (R-Plane)
2
Material Top
1.2
Bottom of Hole
-0.125
Eject slug
Traverse At
Safe Rapid
R-Plane
Modify Feedrate
Modify at hole bottom
Feedrate Modifier (%)
0
Distance from bottom
0
OK
Cancel
Help



Note: When the cut is applied it will look like crosshairs in the circle that was cut.



Section 6

Titan (Using, Creating and Editing styles)

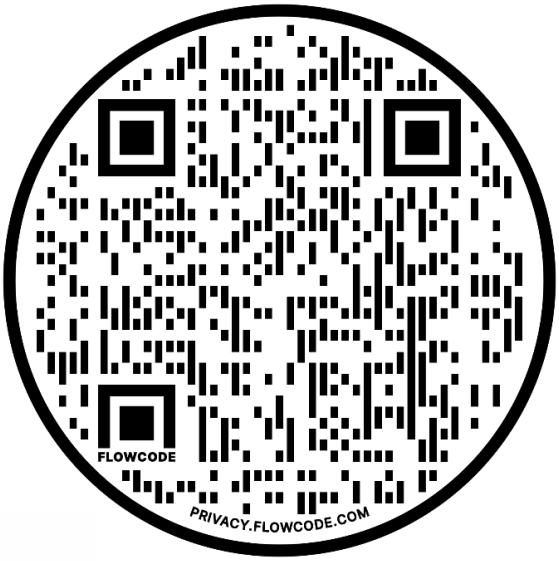
Videos for this section can be found on our website's service and training guides.
https://www.parkindustries.com/stone/service-support/guides/

Suggested videos/guides include

TITAN ALPHACAM MACHINE RIBBON WITH EZ BUTTONS
BASIC TITAN PROGRAMMING (2 VIDEOS+LINKS)
CREATING A STYLE – VIDEO
EDITING A STYLE – VIDEO

Suggested videos/guides from prior sections

USING CUT SHAPE – BASICS (VIDEO+ LINKS)
USING CUT HOLES (VIDEO + LINKS)
USING ADVANCED TOOL DIRECTION METHODS(VIDEO)
EDITING LEAD IN & OUTS (VIDEO)
REDUCING TOOL CHANGES ON THE TITAN (VIDEO)
ADDING A VACUUM POD TO THE TITAN TABLE TEMPLATE (2 VIDEOS)

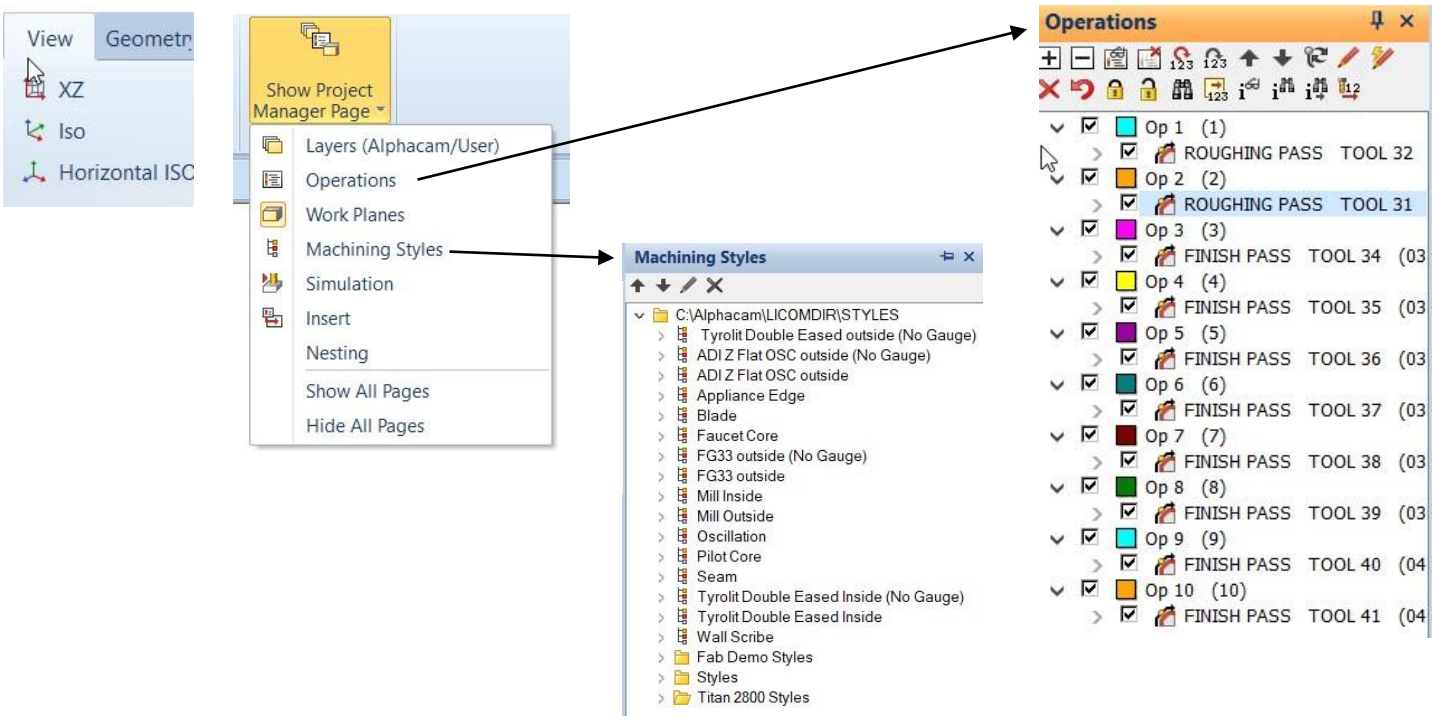




Machining Styles

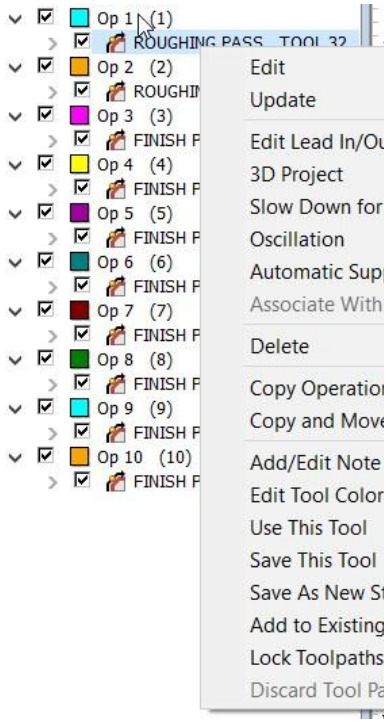
Machining Styles can consist of one or more Cut Paths. Cut Paths must first be created and applied before a Style can be created and saved. All parameters and the order of the cuts are saved into each Style.

The Project Manager windows must open to create and use styles. To open the Project Manager panes select the **View tab**, then choose **Show Project Manager**.



There are normally 4 Project Manager panels are used when programming the Titan: Layers, Operations, Machining Styles, and Simulation. They are docked on the left side of the screen. You can click on a tab to display the one that you need.

Creating a Machining Style



1. Right click on the tool in operations tab
2. Select either to “Save as New Style” or “Add to Existing Style”
3. Enter a name for a new style or select the machining style to add the tool to

1. Apply an Operation (cut) to geometry

a. **Select Tool** from list

b. Insert tool into drawing

c. **Cut Shape**

d. Fill out all parameter windows for cutting properties

e. Choose shape to cut and finish

2. Convert the Operation (cut) to a Style

a. Choose Operation (cut) from the operation window

b. Right click on the Operation (cut) from Operation window and choose “Save as New Style”

c. Choose desired location to save to

i. Default location is C:\AlphaCam\LICOMDIR\STYLES

ii. Style sets may be saved in created folder

Copying a Style

1. **Choose a Style to copy.**
    a. Right click on the desired style from the Machining styles window.
    b. Choose "Copy Style"
    c. Find the "Copy of" in the Machining styles window and rename it to a new style name.
2. **Expand the style to Edit the operations within it.**
    a. Right click on the operation and choose "Edit Operation"
    b. Edit each menu and choose OK as needed.
    c. The changes will automatically be saved into the new style.

Adding to and changing order of Operations (cuts) in a Style

1. Create or Copy/Edit operation

a. To Create a new operation follow the steps in “Creating a Style” until step 2.a. then:
    i. Right click on the cut path (from Operation window) and choose “Add to existing Style”
    ii. Choose desired Style name to save to
b. To Copy/Edit operation
    i. Right click on the cut path (from Operation window) and choose “Copy Operation”

ii. Right click on the Styles name that you wish to paste the op- eration into and choose “Paste Operation”

iii. Right click on the copied cut path (from Operation window) and choose “Edit Operation” iv. Change desired fields from parameter window

2. Change order of operations in the Style

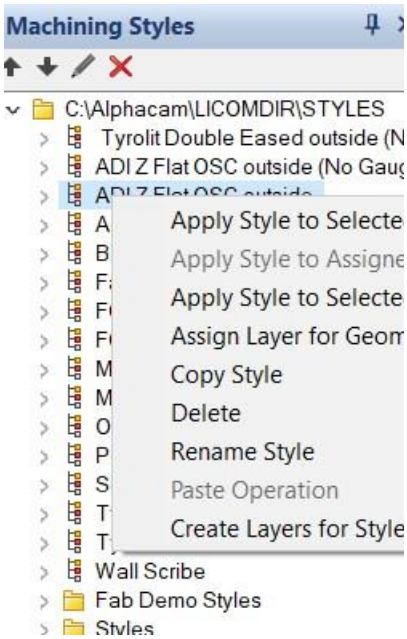
a. Display the Machining Styles tab in Project Manager

b. Expand (+) the Style to view its operations

c. Choose an operation to change the order of

i. Use the red up/down arrow (at top) to arrange the order of each operation within the style

Applying a Machining Style



1. Create the geometry
2. Set tool directions
3. Right click on the machining style- “Apply Style to Selected Geometries”
4. Select geometries and finish

Titan Programming Basic Steps

To program in Alphacam you will need to use many features from the Project Manager (PM) panel. You may need to turn it on by going to View tab then Show Project Manager. The panels you will need include Layer, Operation, Machining Style, and Simulation.

1. Create /fix finished size countertop geometries. *(Reference Pg. 43 for detailed instructions)*
    A. Label parts as needed.
    B. Change old layer to geometry. (If using a .dxf digital template file)
    C. Perform Common Line Removal, Join geometry together.
    D. Fillet any corners if needed.
    E. Insert sinks, faucet holes, cooktops, etc.
    F. Insert/draw seams and use Fabricate Cut to separate parts. Delete original seam geometry.
    G. If there is matching back splash, create it now.
    H. If there are miters, create apron with or without miter properties (depending on if Slabsmith will be used)
    I. **Save as** "*Jobname*" in desired folder. If needed, create a new folder for job files

---

2. Edit-**Offset** the outer perimeter geometry of each part .188" to the *Outside* which will be used as the Saw file and will also be used to set the Material Layer for the Titan. Any Cutouts that will be machined on the Titan which will be cut out completely on the saw need to be offset to the *Inside* .188" for the Saw file.
3. Edit-**Mirror** all parts and fix any Labels as needed. You may place *"Mirrored"* and *"Saw"* labels in the drawing if desired.
4. (Optional) Edit-**Delete** or Edit- **Change** (to a Construction layer) geometries on both the saw and Titan sides that will not be needed for programming.
5. Home-**Save Selected** all the geometries and labels needed for the saw file (Outer perimeter geometry and Inner cutout geometry) and name the file *"Jobname.Slab1".* Then delete the saw side so only the Titan side is visible in the screen.
6. File-**Save As** the drawing as *"Jobname.Titan1".* At this point after saving- go back and program your saw.

---

7. Titan-**Move Parts** away from each other so no outside perimeters are touching.
8. 3D-**Set Materials** set the Material Top in Z to match the material thickness. (Typically, 1.2" for 3CM, 0.8" for 2CM) Material bottom and XY stock should be 0. Check the box to "Delete original geometry". Only chose one perimeter at a time.
9. Project Manager-**Layers** Turn off the "Material Layer" temporarily to ensure it does not get exploded or broke in next step.
10. Edit-**Break** geometry at the end points of common cut edges so they are isolated from each other. Ensure all common cut edges are **Join**ed together. An alternative method is to **Explode** the geometry, and then **Join** back together only the common cut edges.
11. Project Manager-**Layers** Turn the Material Layer back on.
12. File-**Save** the drawing to overwrite the original file. (You are not re-naming the file, just re-saving).

---

13. Machine-**Tool Directions** Set tool directions on all geometries.
    A. For **Open Geometry**:
        1. Direction is **Reverse**, Side is **Left**, and choose **No** in Set Start Point on Closed Geometries.
    B. For **Closed Geometry**:
        1. On *islands* or *Outside* cuts, Direction is **CW,** and side is **Outside** and choose **Manual** in Set Start Point on Closed Geometries.
        2. On *sinks* or *Inside* cuts, Direction is **CCW,** and side is **Inside** and choose **Manual** in Set Start Point on Closed Geometries.
14. Edit-**Move** all parts to the left of the 0,0 to make room for the Titan table template.
15. Titan-**Insert Table Template** from the Titan tab.
16. Titan-**Move Parts** or **Rotate Parts** as needed to place the parts onto the Titan table.

    Make sure that your Material Layers are for sure turned back on
17. Home-**Save Selected** any parts that did not fit on the table into a separate file *"Jobname.Titan2"*. After saving the extra parts, **Delete** them from the (current) drawing.
18. Edit-**Copy, Move,** and/or **Rotate** vacuum pods into place under all parts on the Titan table. **Delete** all extra pods from the template not being used.
    ****Make sure you leave enough room between parts for the lead in and lead out, and make sure you won't have toolpaths outside the machine limits****.*
19. Apply toolpaths using either saved **Machining Styles** or by using **Select Tool** and **Cut Shape**. Apply the toolpaths in the following order, or your desired cutting order.
    1. Coring (Pilot Holes, Faucet Holes).
    2. Milling off excess material (Fingerbit), add Park / Pause Machine if needed.
    3. Wall scribes.
    4. Seams.
    5. Outside Profiles.
    6. Mill sink cutouts, add Park / Pause Machine if needed.
    7. Profile sinks.
20. Project Manager-**Operations** change, order, and group toolpaths and operations as needed.
21. Project Manager-**Simulation** run simulation of your program to verify all toolpaths are correct. Pay attention for tool collisions with other parts, or with vacuum pods and clamp fixtures. Correct any errors.
22. File-**Save** the drawing to overwrite the original file. (You are not re-naming the file, just re-saving).
23. Titan-**Send G-Code** to the machine and fill out and print job sheet if desired.
24. If there were parts that did not fit on the table that were previously saved in step 18, open the file, and start from step 16 and repeat the process.

Titan Programming Fast Track

1. Clean up template or Drawing. **Save as** "*Jobname*" in desired folder.
2. **Offset** .188" for the Saw file.
3. **Mirror** all parts and fix any Labels. You may place *"Mirrored"* and "Saw" labels in the drawing if desired.
4. (optional) **Delete** or **Change** geometries not needed on both the saw and Titan sides.
5. **Save Selected** all the for the saw *"Jobname.S1"* Delete the saw side so only the Titan side is visible
6. **Save As** *"Jobname.Titan1"*. At this point after saving- go back and program your saw.
7. **Move Parts** away from each other so no outside perimeters are touching
8. **Set Materials**. (Typically, 1.2" for 3CM, 0.8" for 2CM).
9. **Layers** Turn off the "Material Layer" temporarily
10. **Break** or **Explode** the geometry, and then **Join** back together only the common cut edges.
11. **Layers** Turn the Material Layer back on.
12. **Save** the drawing to overwrite the original file. (You are not re-naming the file, just re-saving).
13. **Tool Directions** Set tool directions on all geometries.
    A. For **Open Geometry**:
        1. **Reverse**, **Left**, and choose **No**.
    B. For **Closed Geometry**:
        1. For *islands*, Direction is **CW**, **Outside** and choose **Manual**
        2. For *sinks*, Direction is **CCW**, **Inside** and choose **Manual**
14. **Move** all parts to make room for the Titan table template.
15. **Insert Table Template** from the Titan tab.
16. Make sure that your Material Layers are for sure turned back on **Move Parts** or **Rotate Parts**
17. **Save Selected** *"Jobname.Titan2"*, and **Delete**
18. **Copy, Move,** and/or **Rotate** vacuum. **Delete** all extra pods
19. Apply **Machining Styles** or **Select Tool** and **Cut Shape**
    1. Core
    2. Mill relief
    3. Wall scribes.
    4. Seams.
    5. Outside Profiles.
    6. Mill sink cutouts, add Park / Pause Machine if needed.
    7. Profile sinks.
20. **Operations** change, order
21. **Simulation**
22. **Save**
23. **Send G-Code**
24. Finish other programs *("Jobname.Titan2")*

Section 7

Fab Center Saw

Videos for this section can be found on our website's service and training guides.
https://www.parkindustries.com/stone/service-support/guides/

Suggested videos/guides include

SETTING UP A BLADE ON CNC SAWS INCLUDING FOR "ULTRA COMPACT" MATERIALS

PROGRAMMING MITERS ON CNC SAWS IN DEPTH (3 VIDEOS)

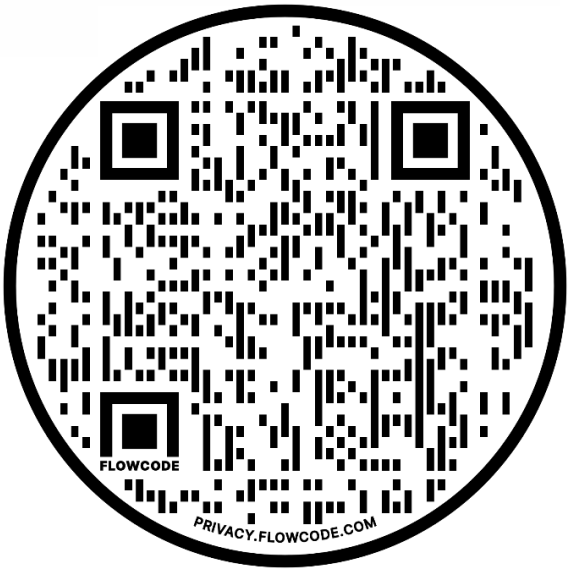
USING PARK AND PAUSE (VIDEO)

STOPPING A MITER BEFORE A SEAM (VIDEO)

SPACING PARTS TO REDUCE SECOND TOOLING OPERATIONS (VIDEO)

ORDERING SAW CUTS FOR IMPROVED PARTS HOLD DOWN (VIDEOS)

SLABSMITH SOFTWARE VIDEO TUTORIAL SERIES

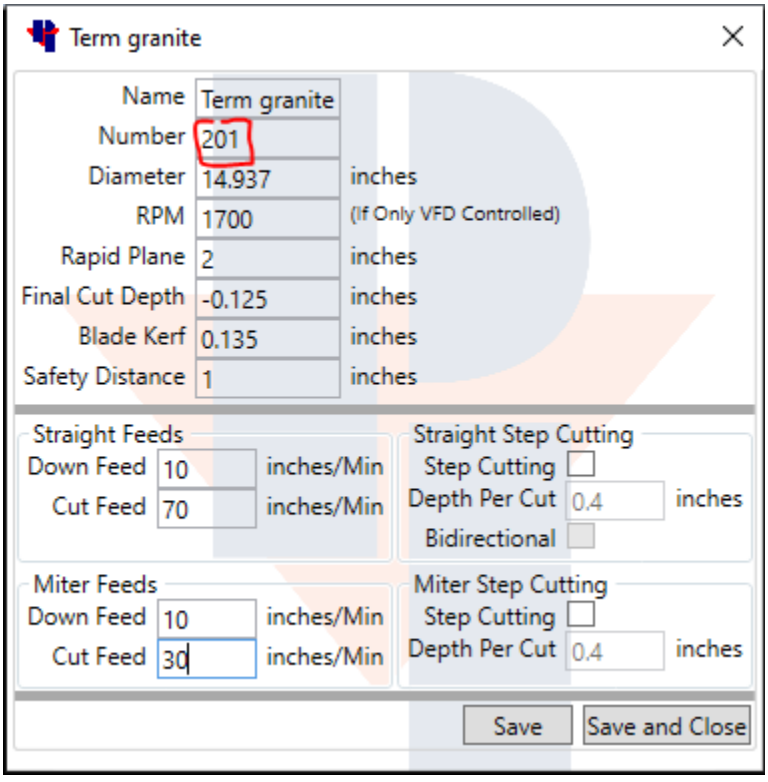




Adding Tool information on the Fab Center with Tool Examples

Setup the tool and parameters for every tool type. Each Fab Center Machine Profile may have up to 20 tools saved in Alphacam, but there can only be ONE BLADE active at a time. You may add tools in the **Machine Profile Setup** window, or you may also add tools in the Titan (Fab Center tab) by pressing the **Tool Select** button.

On the Fab Center - **ALL BLADES** must have a **tool number of 201**



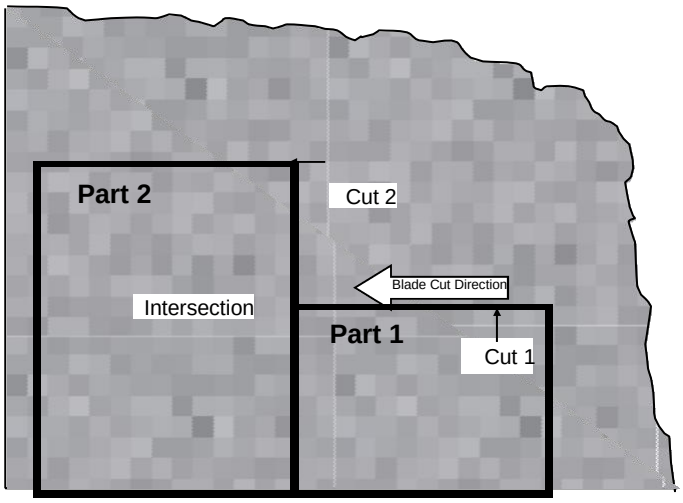
Building the Tool Library for the Saws

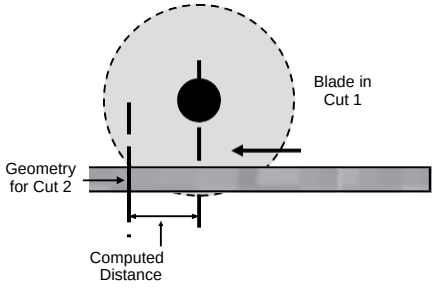
Tools must be entered and all parameters setup in the Tool library in the saw's Park Industries Ribbon before they can be used to create cuts.

Safety Distance detail

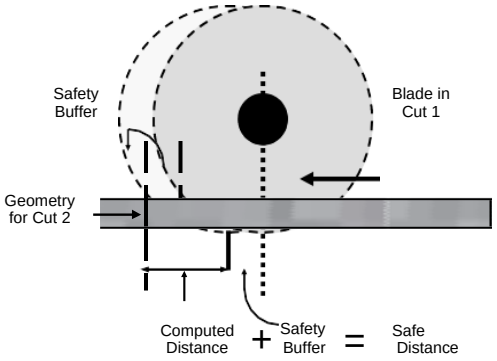
Safe Distance Explanation

In the example below, as the blade finishes cut 1 of part 1 it approaches cut 2 of part 2. The blade motion must stop before reaching the intersection point.





Given the correct tooling parameters, the software computes the distance required to prevent the blade from cutting into an intersecting geometry shown as computed distance in the top figure. If the machine tool parameters match the program tool parameters, and the correct material thickness is used, the blade will always stop before entering the next geometry. But since this does not always happen in the real world, a safety buffer can be used. (See bottom figure).



Safety Distance is a value added to the computed blade over travel stop distance to help prevent the blade from cutting into an intersecting geometry. This compensates for varying stone thickness, blade diameter, and other factors.

**** Important Note:** In the instance of a Voyager, **Tight Cut** does not use **Safety distance!!!**

Fab Center Saw Programming Sequence Without Slabsmith

1. Create/fix finished size countertop geometries. *(Reference Pg. 43 for detailed instructions)*
   - A. Label parts as needed.
   - B. Change old layer to geometry. (If using a .dxf digital template file)
   - C. Perform Common Line Removal, Join geometry together.
   - D. Fillet any corners if needed.
   - E. Insert sinks, faucet holes, cooktops, etc.
   - F. Insert/draw seams and use Fabricate Cut to separate parts. Delete original seam geometry.
   - G. If there is matching back splash, create it now.
   - H. If offset is needed for further machining (i.e., going to a Titan) be sure to start on the Titan check list. Make sure parts are joined before proceeding.
   - I. If there are miters, create aprons and assign other undercuts/ overcuts
   - J. **Save as** "*Jobname*" in desired folder. If needed, create a new folder for job files

---

2. **Save as** file to correct job folder, and re-name appropriately. (i.e.*Jobname.Slab1*) Be sure to eliminate spaces and special characters in the file name for the barcode scanner.

3. **Move** all parts to the left of the 0,0 to make room for the **Titan** table template.

4. **Insert Table Template** from the **Titan** tab.

5. Draw an accurate border of the slab being cut. Be sure the slab origin point is at 0,0 location. Make sure to **Set Materials** on the outside of the slab for the laser.

6. **Move** and/or **Rotate** all parts inside the slab border using either the Park Industries **Titan** tool bar manipulation commands, or the Alphacam Edit commands. You may also use the **Join Parts-Common Line Cuts** command if needed or applicable.

7. Use **Home-Save Selected** to save any parts that did not fit on the slab/ **Titan** table into a separate file *(Jobname.Slab2)*. After saving, **Delete** all the parts that were just previously saved in the new file.

8. **Automatic Tool Path** Verify all settings are correct, then select parts only, and then finish.

9. Apply any needed **Manual cuts** and **Delete** or modify any toolpaths as necessary. Be sure that you don't have cuts violating any boundaries.

10. **Extend cuts** to the slab border and change **cooktop depths** as needed.

11. **Order of cuts** and operations as needed.

12. **Send G Code** to Machine using the icon on the **Titan** tool bar. (This also re-saves the file.)

13. If there are any uncut parts, (any files that were saved in step #7) open file and start from step #4.

Fab Center Saw Programming Sequence With Slabsmith

1. Create/fix finished size countertop geometries. *(Reference Pg. 43 for detailed instructions)*
   A. Label parts as needed.
   B. Change old layer to geometry. (If using a .dxf digital template file)
   C. Perform Common Line Removal, Join geometry together.
   D. Fillet any corners if needed.
   E. Insert sinks, faucet holes, cooktops, etc.
   F. Insert/draw seams and use Fabricate Cut to separate parts. Delete original seam geometry.
   G. If there is matching back splash, create it now.
   H. If offset is needed for further machining (i.e., going to a Titan) be sure to start on the Titan check list. Make sure parts are joined before proceeding.
   I. If there are miters, create aprons as rectangles, (Backsplash Button) and **do not** apply Miter properties until after the Slabsmith .dxf layout is input back into Alphacam for programming.
   J. **Save as** "*Jobname*" in desired folder. If needed, create a new folder for job files

---

2. Go to **Home** → **Output CAD** and save the DXF as "*Jobname.dxf* for Slabsmith in the correct job folder.
3. Open *Slabsmith*, and then open a new session of *Perfect Match.* Open the DXF file created in Step #3 and load the slab(s) needed for the current job.
4. Move parts on slabs. **Move**, **rotate**, and\or use the **Common Line, Common line+, or Snaps** (from the Slabsmith toolbar) to do so. Define any remnants if needed (**Reserved Areas**).
5. Once you have the final layout, (customer approved if needed) be sure to save the Layout in Slabsmith. Save the DXF file to the Job folder created in step #2. Be sure to name the files appropriately.

---

6. Go back to Alphacam and input **DXF** file from *Slabsmith*. Go to **Home** → **Input CAD** and select appropriate file. Keep in mind there may be multiple files if multiple slabs were used.
7. Perform the **Common line Removal** command on all parts and choose yes to "Join Resulting Geometries". Make sure to use a selection window around ALL parts.
8. Apply any miter properties if needed (make sure in Auto Toolpath the correct thickness is selected)
9. **Insert Table Template** from the **Titan** tab and be sure that the slab border is against the correct table stops on the table template. Make sure to **Set Materials** on the perimeter of the slab for the laser.
10. **Save as** file to correct job folder, and re-name appropriately. (*Jobname.SlabID#*) Be sure to eliminate spaces and special characters in the file name for the barcode scanner.
11. **Automatic Tool Path** Verify all settings are correct, then select parts only, and then finish.
12. Apply any needed **Manual cuts** and **Delete** or modify any toolpaths as necessary. Be sure that you don't have cuts violating any boundaries.
13. **Extend cuts** to the slab border and change **cooktop depths** as needed.
14. **Order of cuts** and operations as needed.
15. **Send G Code** to Machine using the icon on the **Titan** toolbar. (This also re-saves the file.)
16. If there are any additional DXF files from *Slabsmith* for the job, go back and repeat from step #6.

Section 8

Voyager Saw

Videos for this section can be found on our website's service and training guides.
https://www.parkindustries.com/stone/service-support/guides/

Suggested videos/guides include

VOYAGER XP ALPHACAM MACHINE RIBBON WITH EZ BUTTONS

BASIC VOYAGER XP PROGRAMMING

SETTING UP A BLADE ON CNC SAWS INCLUDING FOR "ULTRA COMPACT" MATERIALS

PROGRAMMING MITERS ON CNC SAWS IN DEPTH (3 VIDEOS)

ADDING A TIGHT CUT – VOYAGER (VIDEO)

USING PARK AND PAUSE (VIDEO)

STOPPING A MITER BEFORE A SEAM (VIDEO)

SPACING PARTS TO REDUCE SECOND TOOLING OPERATIONS (VIDEO)

ORDERING SAW CUTS FOR IMPROVED PARTS HOLD DOWN (VIDEOS)

SLABSMITH SOFTWARE VIDEO TUTORIAL SERIES

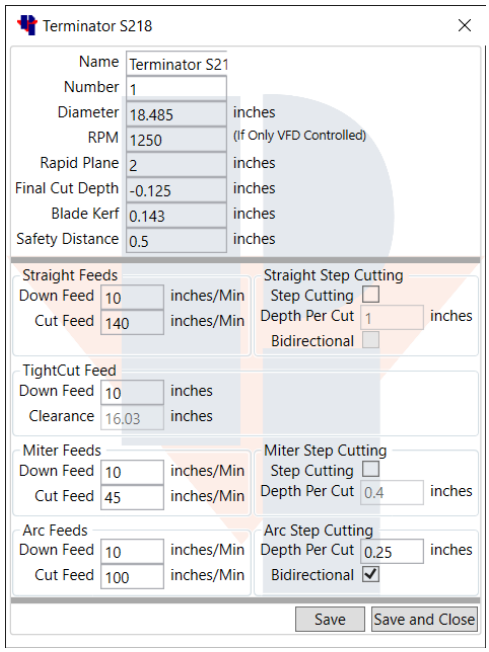




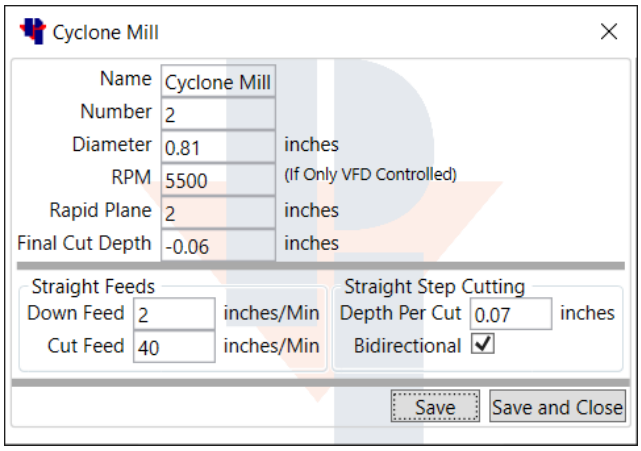
Voyager Tool Examples

Setup the tool and parameters for every tool type. Each Voyager Machine Profile may have up to 20 tools saved in Alphacam, but there can only be ONE BLADE, ONE MILL BIT and ONE CORE BIT active at a time. You may add tools in the **Machine Profile Setup** window, or you may also add tools in the **Voyager** tab by pressing the **Tool Select** button. Tool numbers must match the Voyager's tool library on the machine.

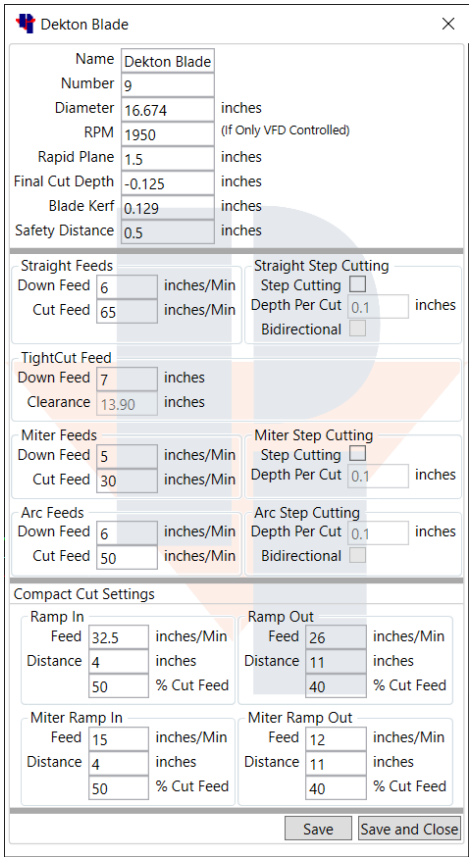
Blade



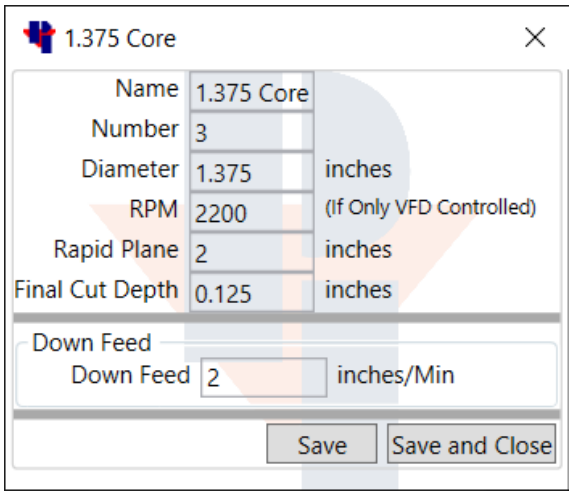
Mill Bit



Ultra Compact Blade



Core Bit



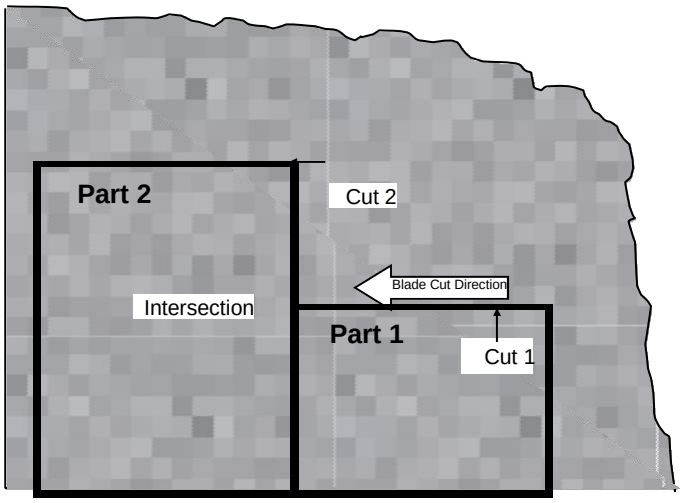
Building the Tool Library for the Saws

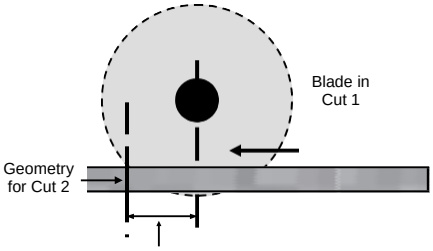
Tools must be entered and all parameters setup in the Tool library in the saw's Park Industries Ribbon before they can be used to create cuts.

Safety Distance detail

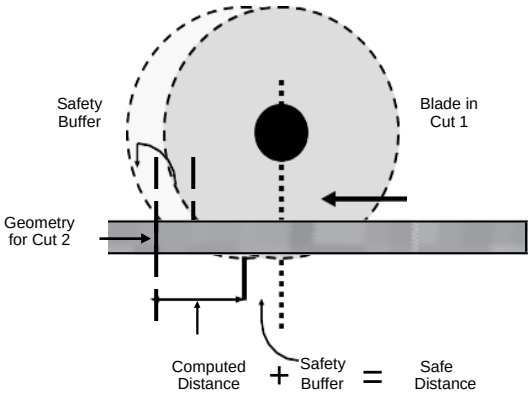
Safe Distance Explanation

In the example below, as the blade finishes cut 1 of part 1 it approaches cut 2 of part 2. The blade motion must stop before reaching the intersection point.





Given the correct tooling parameters, the software computes the distance required to prevent the blade from cutting into an intersecting geometry shown as computed distance in the top figure. If the machine tool parameters match the program tool parameters, and the correct material thickness is used, the blade will always stop before entering the next geometry. But since this does not always happen in the real world, a safety buffer can be used. (See bottom figure).



Safety Distance is a value added to the computed blade over travel stop distance to help prevent the blade from cutting into an intersecting geometry. This compensates for varying stone thickness, blade diameter, and other factors.

**** Important Note:** In the instance of a Voyager, **Tight Cut** does not use **Safety distance!!!**

Computed Distance

VOYAGER Programming Sequence Without Slabsmith

1. Create/fix finished size countertop geometries. *(Reference Pg. 43 for detailed instructions)*
    a. Label parts as needed.
    b. Change old layer to geometry. (If using a .dxf digital template file)
    c. Perform Common Line Removal, Join geometry together.
    d. Fillet any corners if needed.
    e. Insert sinks, faucet holes, cooktops, etc.
    f. Insert/draw seams and use Fabricate Cut to separate parts. Delete original seam geometry.
    g. If there is matching back splash, create it now.
    h. If offset is needed for further machining (i.e., going to a Titan) be sure to start on the Titan check list. Make sure parts are joined before proceeding.
    i. If there are miters, create aprons and assign other undercuts/ overcuts
    j. **Save as** "*Jobname*" in desired folder. If needed, create a new folder for job files

---

2. **Save as** file to correct job folder, and re-name appropriately. (i.e.*Jobname.Slab1*) Be sure to eliminate spaces and special characters in the file name for the barcode scanner.
3. **Move** all parts to the left of the 0,0 to make room for the Voyager table template.
4. **Insert Table Template** from the Voyager tab.
5. Draw an accurate border of the slab being cut. Be sure the slab origin point is at 0,0 location.
6. **Move** and/or **Rotate** all parts inside the slab border using either the Park Industries Voyager tool bar manipulation commands, or the Alphacam Edit commands. You may also use the **Join Parts-Common Line Cuts** command if needed or applicable.
7. Use **Home-Save Selected** to save any parts that did not fit on the slab/Voyager table into a separate file *(Jobname.Slab2)*. After saving, **Delete** all the parts that were just previously saved in the new file.
8. **Automatic Tool Path** Verify all settings are correct, then select parts only, and then finish.
9. Apply any needed **Manual cuts** and **Delete** or modify any toolpaths as necessary. Be sure that you don't have cuts violating any boundaries.
10. **Extend cuts** to the slab border and change **cooktop depths** as needed.
11. **Order of cuts** and operations as needed. (1.Core holes, 2.Incremental routing, 3.Blade cuts, etc.)
12. **Send G Code** to Machine using the icon on the **Voyager** tool bar. (This also re-saves the file.
13. If there are any uncut parts, (any files that were saved in step #7) open file and start from step #4.

VOYAGER Programming Sequence With Slabsmith

1. Create/fix finished size countertop geometries. *(Reference Pg. 43 for detailed instructions)*
   a. Label parts as needed.
   b. Change old layer to geometry. (If using a .dxf digital template file)
   c. Perform Common Line Removal, Join geometry together.
   d. Fillet any corners if needed.
   e. Insert sinks, faucet holes, cooktops, etc.
   f. Insert/draw seams and use Fabricate Cut to separate parts. Delete original seam geometry.
   g. If there is matching back splash, create it now.
   h. If offset is needed for further machining (i.e., going to a Titan) be sure to start on the Titan check list. Make sure parts are joined before proceeding..
   i. If there are miters, create aprons as rectangles, (Backsplash Button) and **do not** apply Miter properties until after the Slabsmith .dxf layout is input back into Alphacam for programming.
   j. **Save as** "*Jobname*" in desired folder. If needed, create a new folder for job files

---

2. Go to **Home** → **Output CAD** and save the DXF as "*Jobname.dxf* for Slabsmith in the correct job folder.
3. Open *Slabsmith*, and then open a new session of *Perfect Match.* Open the DXF file created in Step #3 and load the slab(s) needed for the current job.
4. Move parts on slabs. **Move**, **rotate**, and\or use the **Common Line, Common line+, or Snaps** (from the Slabsmith toolbar) to do so. Define any remnants if needed (**Reserved Areas**).
5. Once you have the final layout, (customer approved if needed) be sure to save the Layout in Slabsmith. Save the DXF file to the Job folder created in step #2. Be sure to name the files appropriately.

---

6. Go back to Alphacam and input **DXF** file from *Slabsmith*. Go to **Home** → **Input CAD** and select appropriate file. Keep in mind there may be multiple files if multiple slabs were used.
7. Perform the **Common line Removal** command on all parts and choose yes to "Join Resulting Geometries". Make sure to use a selection window around ALL parts.
8. Apply any miter properties if needed (make sure in Auto Toolpath the correct thickness is selected)
9. **Insert Table Template** from the **Voyager** tab and be sure that the slab border is against the correct table stops on the table template.
10. **Save as** file to correct job folder, and re-name appropriately. (*Jobname.SlabID#*) Be sure to eliminate spaces and special characters in the file name for the barcode scanner.
11. **Automatic Tool Path** Verify all settings are correct, then select parts only, and then finish.
12. Apply any needed **Manual cuts** and **Delete** or modify any toolpaths as necessary. Be sure that you don't have cuts violating any boundaries.
13. **Extend cuts** to the slab border and change **cooktop depths** as needed.
14. **Order of cuts** and operations as needed. (1.Core holes, 2.Incremental routing, 3.Blade cuts, etc.)
15. **Send G Code** to Machine using the icon on the **Voyager** toolbar. (This also re-saves the file.)
16. If there are any additional DXF files from *Slabsmith* for the job, go back and repeat from step #6.

Section 9

SABERjet and Javelin Saws

Videos for this section can be found on our website's service and training guides.
https://www.parkindustries.com/stone/service-support/guides/

Suggested videos/guides include

SABERJET ALPHACAM MACHINE RIBBON WITH EZ BUTTONS

BASIC SABERJET AND SABERJET XP PROGRAMMING

SETTING UP A BLADE ON CNC SAWS INCLUDING FOR "ULTRA COMPACT" MATERIALS

PROGRAMMING MITERS ON CNC SAWS IN DEPTH (3 VIDEOS)

USING PARK AND PAUSE (VIDEO)

STOPPING A MITER BEFORE A SEAM (VIDEO)

SPACING PARTS TO REDUCE SECOND TOOLING OPERATIONS (VIDEO)

ORDERING SAW CUTS FOR IMPROVED PARTS HOLD DOWN (VIDEOS)

SLABSMITH SOFTWARE VIDEO TUTORIAL SERIES



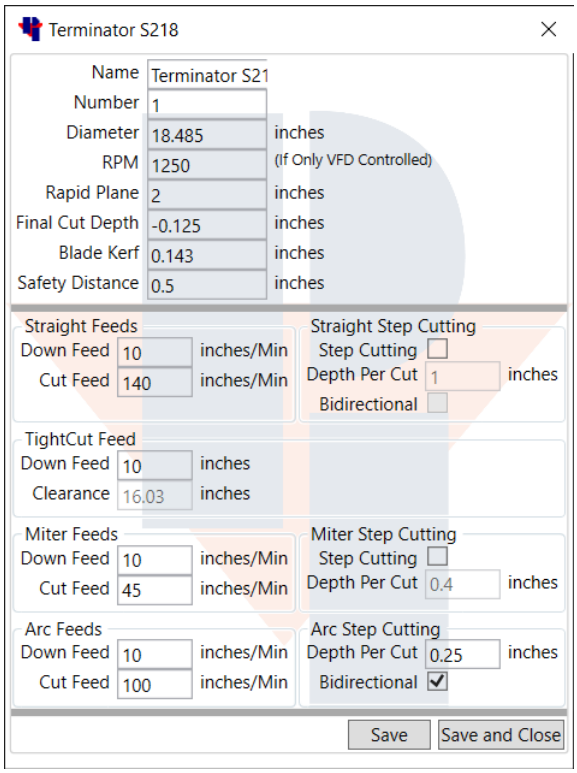
SERVICE & TRAINING GUIDES

Adding Tool information on the SABERjet or Javelin with Tool Examples

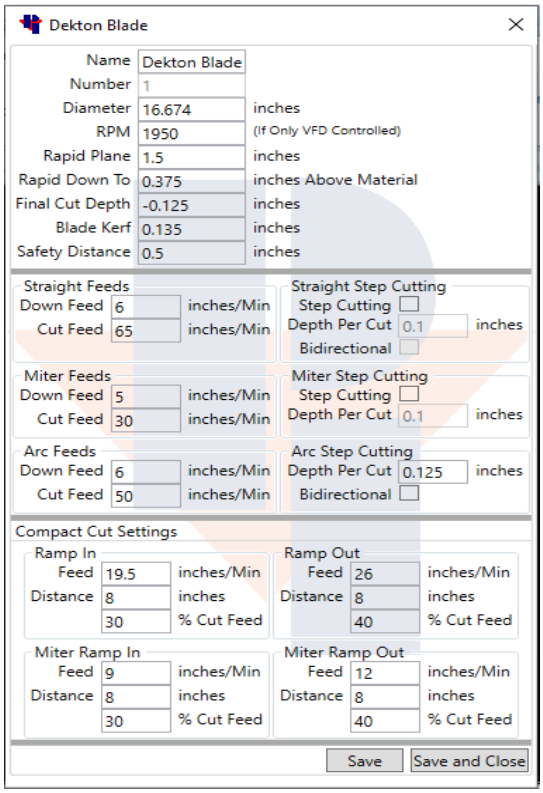
Setup the tool and parameters for every tool type **(do not use special characters in tool name)**. Each saw's Machine Profile may have up to 20 tools saved in Alphacam, but there can only be ONE BLADE and ONE WATERJET active at a time. You may add tools in the **Machine Profile Setup** window, or you may also add tools in the **saw's** tab by pressing the **Tool Select** button.

On these saws - **ALL BLADES** must have a **tool number of 1**

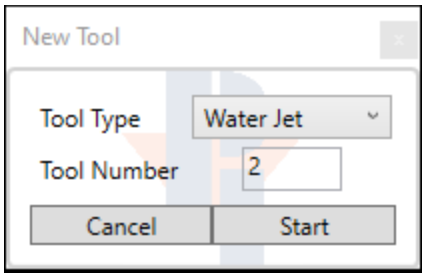
Blade

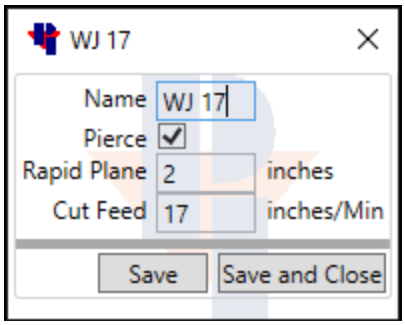


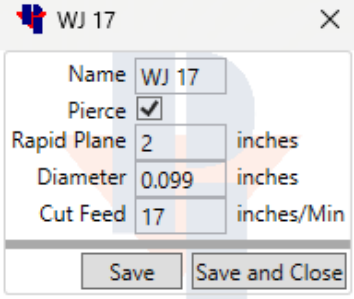
Ultra Compact Blade



On these saws - **ALL Waterjets** must have a **tool number of 2**







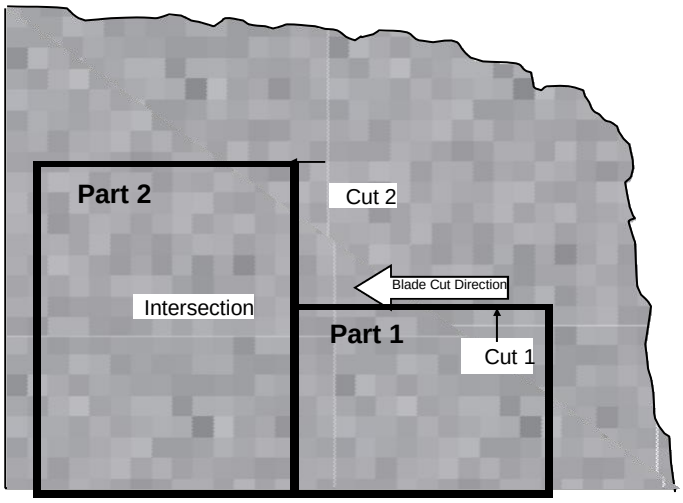
Building the Tool Library for the Saws

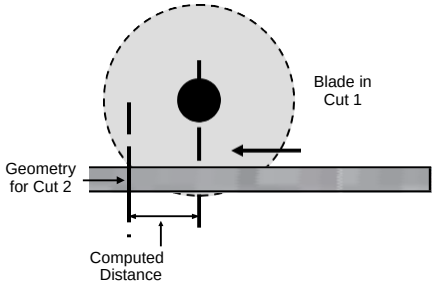
Tools must be entered and all parameters setup in the Tool library in the saw's Park Industries Ribbon before they can be used to create cuts.

Safety Distance detail

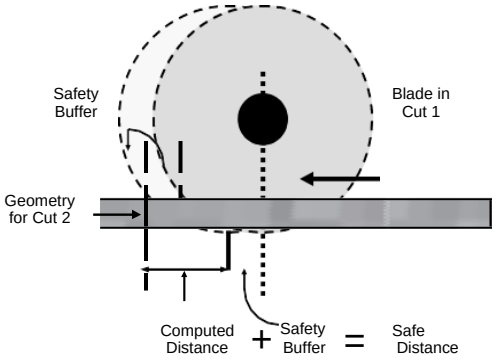
Safe Distance Explanation

In the example below, as the blade finishes cut 1 of part 1 it approaches cut 2 of part 2. The blade motion must stop before reaching the intersection point.





Given the correct tooling parameters, the software computes the distance required to prevent the blade from cutting into an intersecting geometry shown as computed distance in the top figure. If the machine tool parameters match the program tool parameters, and the correct material thickness is used, the blade will always stop before entering the next geometry. But since this does not always happen in the real world, a safety buffer can be used. (See bottom figure).



Safety Distance is a value added to the computed blade over travel stop distance to help prevent the blade from cutting into an intersecting geometry. This compensates for varying stone thickness, blade diameter, and other factors.

**** Important Note:** In the instance of a Voyager, **Tight Cut** does not use **Safety distance!!!**

SABERjet and Javelin Programming Sequence Without Slabsmith

1. Create/fix finished size countertop geometries. *(Reference Pg. 43 for detailed instructions)*
   A. Label parts as needed.
   B. Change old layer to geometry. (If using a .dxf digital template file)
   C. Perform Common Line Removal, Join geometry together.
   D. Fillet any corners if needed.
   E. Insert sinks, faucet holes, cooktops, etc.
   F. Insert/draw seams and use Fabricate Cut to separate parts. Delete original seam geometry.
   G. If there is matching back splash, create it now.
   H. If offset is needed for further machining (i.e., going to a Titan) be sure to start on the Titan check list. Make sure parts are joined before proceeding.
   I. If there are miters, create aprons and assign other undercuts/ overcuts
   J. **Save as** "*Jobname*" in desired folder. If needed, create a new folder for job files

---

2. **Save as** file to correct job folder, and re-name appropriately. (i.e.*Jobname.Slab1*) Be sure to eliminate spaces and special characters in the file name for the barcode scanner.

3. **Move** all parts to the left of the 0,0 to make room for the **Saw's** table template.

4. **Insert Table Template** from the **Saw's** tab.

5. Draw an accurate border of the slab being cut. Be sure the slab origin point is at 0,0 location, *or if needed, against the Longer "Miter" stops (specific to the **Javelin**).*

6. **Move** and/or **Rotate** all parts inside the slab border using either the Park Industries EZ tool bar manipulation commands, or the Alphacam Edit commands. You may also use the **Join Parts-Common Line Cuts** command if needed or applicable.

7. Use **Home-Save Selected** to save any parts that did not fit on the slab/ **saw's** table into a separate file *(Jobname.Slab2)*. After saving, **Delete** all the parts that were just previously saved in the new file.

8. **Automatic Tool Path** Verify all settings are correct, then select parts only, and then finish.

9. Apply any needed **Manual cuts** and **Delete** or modify any toolpaths as necessary. Be sure that you don't have cuts violating any boundaries.

10. **Extend cuts** to the slab border and change **cooktop depths** as needed.

11. **Order of cuts** and operations as needed.

12. **Send G Code** to Machine using the icon on the **Saw's** tool bar. (This also re-saves the file.)

13. If there are any uncut parts, (any files that were saved in step #7) open file and start from step #4.

SABERjet and Javelin Programming Sequence With Slabsmith

1. Create/fix finished size countertop geometries. *(Reference Pg. 43 for detailed instructions)*
   A. Label parts as needed.
   B. Change old layer to geometry. (If using a .dxf digital template file)
   C. Perform Common Line Removal, Join geometry together.
   D. Fillet any corners if needed.
   E. Insert sinks, faucet holes, cooktops, etc.
   F. Insert/draw seams and use Fabricate Cut to separate parts. Delete original seam geometry.
   G. If there is matching back splash, create it now.
   H. If offset is needed for further machining (i.e., going to a Titan) be sure to start on the Titan check list. Make sure parts are joined before proceeding.
   I. If there are miters, create aprons as rectangles, (Backsplash Button) and **do not** apply Miter properties until after the Slabsmith .dxf layout is input back into Alphacam for programming.
   J. **Save as** "*Jobname*" in desired folder. If needed, create a new folder for job files.

---

2. Go to **Home** → **Output CAD** and save the DXF as *Jobname.dxf* for Slabsmith in the correct job folder.
3. Open *Slabsmith*, and then open a new session of *Perfect Match.* Open the DXF file created in Step #3 and load the slab(s) needed for the current job.
4. Move parts on slabs. **Move**, **rotate**, and\or use the **Common Line, Common line+, or Snaps** (from the Slabsmith toolbar) to do so. Define any remnants if needed (**Reserved Areas**).
5. Once you have the final layout, (customer approved if needed) be sure to save the Layout in Slabsmith. Save the DXF file to the Job folder created in step #2. Be sure to name the files appropriately. Output using the correct origin for the slab, either 0,0 (or the longer miter stops, specific to the **Javelin**).

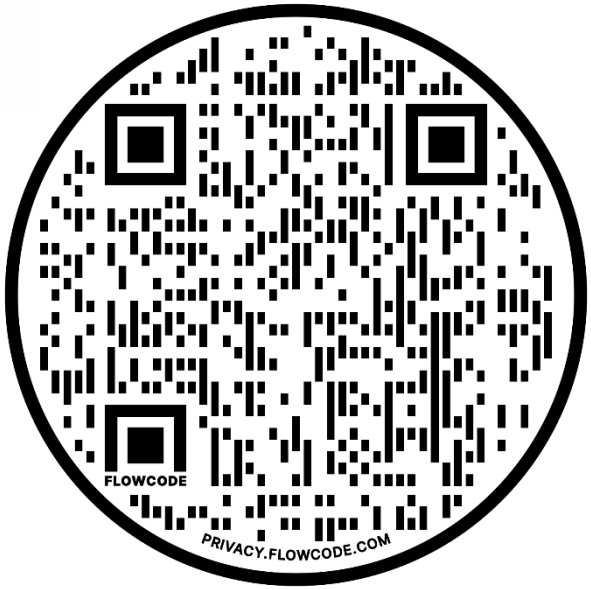
---

6. Go back to Alphacam and input **DXF** file from *Slabsmith*. Go to **Home** → **Input CAD** and select appropriate file. Keep in mind there may be multiple files if multiple slabs were used.
7. Perform the **Common line Removal** command on all parts and choose yes to "Join Resulting Geometries". Make sure to use a selection window around ALL parts.
8. Apply any miter properties if needed (make sure in Auto Toolpath the correct thickness is selected)
9. **Insert Table Template** from the **saw's** tab and be sure that the slab border is against the correct table stops on the table template.
10. **Save as** file to correct job folder, and re-name appropriately. (*Jobname.SlabID#*) Be sure to eliminate spaces and special characters in the file name for the barcode scanner.
11. **Automatic Tool Path** Verify all settings are correct, then select parts only, and then finish.
12. Apply any needed **Manual cuts** and **Delete** or modify any toolpaths as necessary. Be sure that you don't have cuts violating any boundaries.
13. **Extend cuts** to the slab border and change **cooktop depths** as needed.
14. **Order of cuts** and operations as needed.
15. **Send G Code** to Machine using the icon on the **saw's** toolbar. (This also re-saves the file.)
16. If there are any additional DXF files from *Slabsmith* for the job, go back and repeat from step #6.

Section 10

Note Pages & Pull-outs

Many other videos and guides exist on our website's service and training guides.
https://www.parkindustries.com/stone/service-support/guides/



SERVICE & TRAINING GUIDES

Inserting a Clean Sink File

Create your parts or open an existing drawing in Alphacam.

1. **Offset** the front edge to the inside of your part to create the sink *Setback line* where the front edge of the sink will be placed *(commonly 4")* using the *Line / Arc* option in the **Offset** command.
2. Draw a **Line** or use **Offset** to create a sink *Centerline* that will be used to accurately place the sink in the correct spot on your part.
3. Go to the **Home** tab and choose **Insert**. You will then browse on your computer for the correct sink file that you desire and open it. *(This would be a sink drawing that you have previously cleaned up and saved in Alphacam. See sink editing procedure pg. 40-43.)*
4. If you saved your sink correctly at the 0,0 location you will be holding the sink at the base centerline with your curser. Snap it to the **Intersection (F9)** of the *Setback line* and the *Centerline* you previously created.
5. Measure the *Angle* of the *Setback line* using the **Distance / Angle** function in Alphacam. Use your **End Point (F6)** snap key and snap each end of the line measuring in a positive direction. If the angle measures 0, 180, or 360 degrees then you do not need to rotate the sink if it has the correct orientation inside your part. If the angle is not any of those degree values, or if the sink is not orientated correctly you will need to **Rotate** the sink, so that it becomes *Parallel* with the front edge. You will need to proceed to step 6.
6. *(Perform these steps to Rotate the sink only if needed)* Choose the **Rotate** command from the *Edit* tab and either select the *Previous* button on the bottom of the screen or select only the sink and faucet holes as needed and then *Finish*. When prompted to pick a *Base Point* press the **Intersection (F9)** snap key and then select the *Setback line* and the *Centerline* that were originally used to snap the sink to.

   Once the sink is pivoting on the base point you are prompted to choose the *Rotation Angle*. You will then press the **Endpoint (F6)** snap key and select the correct side of the *Setback line* to snap the sink at the correct angle and correct orientation inside your part. When prompted for *How Many Copies* select 1 and press OK, and then *Finish* the command.

7. **Delete** the *Setback line* and the sink *Centerline*.

Titan Programming Basic Steps

To program in Alphacam you will need to use many features from the Project Manager (PM) panel. You may need to turn it on by going to View tab then Show Project Manager. The panels you will need include Layer, Operation, Machining Style, and Simulation.

1. Create /fix finished size countertop geometries. *(Reference Pg. 43 for detailed instructions)*
   A. Label parts as needed.
   B. Change old layer to geometry. (If using a .dxf digital template file)
   C. Perform Common Line Removal, Join geometry together.
   D. Fillet any corners if needed.
   E. Insert sinks, faucet holes, cooktops, etc.
   F. Insert/draw seams and use Fabricate Cut to separate parts. Delete original seam geometry.
   G. If there is matching back splash, create it now.
   H. If there are miters, create apron with or without miter properties (depending on if Slabsmith will be used)
   I. **Save as** "*Jobname*" in desired folder. If needed, create a new folder for job files

---

2. Edit-**Offset** the outer perimeter geometry of each part .188" to the *Outside* which will be used as the Saw file and will also be used to set the Material Layer for the Titan. Any Cutouts that will be machined on the Titan which will be cut out completely on the saw need to be offset to the *Inside* .188" for the Saw file.
3. Edit-**Mirror** all parts and fix any Labels as needed. You may place *"Mirrored"* and *"Saw"* labels in the drawing if desired.
4. (Optional) Edit-**Delete** or Edit- **Change** (to a Construction layer) geometries on both the saw and Titan sides that will not be needed for programming.
5. Home-**Save Selected** all the geometries and labels needed for the saw file (Outer perimeter geometry and Inner cutout geometry) and name the file *"Jobname.Slab1"*. Then delete the saw side so only the Titan side is visible in the screen.
6. File-**Save As** the drawing as *"Jobname.Titan1"*. At this point after saving- go back and program your saw.

Titan Programming Basic Steps

To program in Alphacam you will need to use many features from the Project Manager (PM) panel. You may need to turn it on by going to View tab then Show Project Manager. The panels you will need include Layer, Operation, Machining Style, and Simulation.

1. Create /fix finished size countertop geometries. *(Reference Pg. 43 for detailed instructions)*
   A. Label parts as needed.
   B. Change old layer to geometry. (If using a .dxf digital template file)
   C. Perform Common Line Removal, Join geometry together.
   D. Fillet any corners if needed.
   E. Insert sinks, faucet holes, cooktops, etc.
   F. Insert/draw seams and use Fabricate Cut to separate parts. Delete original seam geometry.
   G. If there is matching back splash, create it now.
   H. If there are miters, create apron with or without miter properties (depending on if Slabsmith will be used)
   I. **Save as** "*Jobname*" in desired folder. If needed, create a new folder for job files

---

2. Edit-**Offset** the outer perimeter geometry of each part .188" to the *Outside* which will be used as the Saw file and will also be used to set the Material Layer for the Titan. Any Cutouts that will be machined on the Titan which will be cut out completely on the saw need to be offset to the *Inside* .188" for the Saw file.
3. Edit-**Mirror** all parts and fix any Labels as needed. You may place *"Mirrored"* and *"Saw"* labels in the drawing if desired.
4. (Optional) Edit-**Delete** or Edit- **Change** (to a Construction layer) geometries on both the saw and Titan sides that will not be needed for programming.
5. Home-**Save Selected** all the geometries and labels needed for the saw file (Outer perimeter geometry and Inner cutout geometry) and name the file *"Jobname.Slab1"*. Then delete the saw side so only the Titan side is visible in the screen.
6. File-**Save As** the drawing as *"Jobname.Titan1"*. At this point after saving- go back and program your saw.

Notes:

F1	F2	F3	F4		F5	F6	F7	F8		F9	F10	F11	F12
Unknown/ Help	Various	Ortho On/Off	Close & Finish		Quadrant	End of	Middle of	Center of		Intersect of	Tangent to	Perpend to	Parallel to

F1	F2	F3	F4		F5	F6	F7	F8		F9	F10	F11	F12
Unknown/ Help	Various	Ortho On/Off	Close & Finish		Quadrant	End of	Middle of	Center of		Intersect of	Tangent to	Perpend to	Parallel to

F1	F2	F3	F4		F5	F6	F7	F8		F9	F10	F11	F12
Unknown/ Help	Various	Ortho On/Off	Close & Finish		Quadrant	End of	Middle of	Center of		Intersect of	Tangent to	Perpend to	Parallel to

F1	F2	F3	F4		F5	F6	F7	F8		F9	F10	F11	F12
Unknown/ Help	Various	Ortho On/Off	Close & Finish		Quadrant	End of	Middle of	Center of		Intersect of	Tangent to	Perpend to	Parallel to

F1	F2	F3	F4		F5	F6	F7	F8		F9	F10	F11	F12
Unknown/ Help	Various	Ortho On/Off	Close & Finish		Quadrant	End of	Middle of	Center of		Intersect of	Tangent to	Perpend to	Parallel to

F1	F2	F3	F4		F5	F6	F7	F8		F9	F10	F11	F12
Unknown/ Help	Various	Ortho On/Off	Close & Finish		Quadrant	End of	Middle of	Center of		Intersect of	Tangent to	Perpend to	Parallel to

Keyboard Shortcuts

Geometry		View		Action	Hotkey
Arc	A	3-D	V	3-D	V
Circle	Ctrl+C	Ghost Tools	Ctrl+G	Add-Ins	`
Common Line Removal	Shift+R	Zoom All	Z	Apply Style	Shift+S
Enclosing Rectangle	Alt+R	Zoom Previous	Ctrl+B	Arc	A
Line	L	Zoom Window	W	Break	B
Rectangle	R			Change	Shift+C
Text	Ctrl+T	**CAD**		Circle	Ctrl+C
		Dimension	Ctrl+D	Clear Memory	Ctrl+Delete
Edit		Distance/Angle	U	Common Line Removal	Shift+R
Break	B	Hide Parts	H	Copy	C
Change	Shift+C	Show All Parts	Alt+H	Cut	Alt+C
Copy	C	User Layers	Ctrl+U	Delete	D
Cut	Alt+C			Dimension	Ctrl+D
Delete	D	**File**		Distance/Angle	U
Explode	X	New	Ctrl+N	Enclosing Rectangle	Alt+R
Extend	E	Open	Ctrl+O	Explode	X
Extend By Distance	Ctrl+E	Save	Ctrl+S	Extend	E
Fillet	F			Extend By Distance	Ctrl+E
Join	J	**Add-Ins/Macros**		Fillet	F
Keep	K	Add-Ins	`	Ghost Tools	Ctrl+G
Mirror	I			Input CAD	Ctrl+I
Move	M	**Machine**		Insert	Shift+I
Offset	O	Apply Style	Shift+S	Join	J
Redo	Ctrl+Y	Tool Direction	Shift+T	Keep	K
Rotate	Ctrl+R			Line	L
Scale	S	**3D**		Mirror	I
Trim	T	Project 3D to 2D	P	Move	M
Undo	Ctrl+Z	Set Material	Shift+M	New	Ctrl+N
				Offset	O
Home				Open	Ctrl+O
Clear Memory	Ctrl+Delete			Project 3D to 2D	P
Input CAD	Ctrl+I			Rectangle	R
Insert	Shift+I			Redo	Ctrl+Y
				Rotate	Ctrl+R
				Save	Ctrl+S
				Scale	S
				Set Material	Shift+M
				Text	Ctrl+T
				Tool Direction	Shift+T
				Trim	T
				Undo	Ctrl+Z
				User Layers	Ctrl+U
				Zoom All	Z
				Zoom Previous	Ctrl+B
				Zoom Window	W
				Hide Parts	H
				Show All Parts	Alt+H

Quick Reference

2CM= .787 inches

3CM= 1.181 inches

Decimal/Metric Conversion

INCHES		mm
1/64	0.015	0.396
1/32	0.031	0.793
3/64	0.046	1.190
1/16	0.062	1.587
5/54	0.078	1.984
3/32	0.093	2.381
7/64	0.109	2.778
1/8	0.125	3.175
9/64	0.140	3.571
5/32	0.156	3.968
11/64	0.171	4.365
3/16	0.187	4.762
13/64	0.203	5.159
7/32	0.218	5.556
15/64	0.234	5.953
1/4	0.250	6.350
17/64	0.265	6.746
9/32	0.281	7.143
19/64	0.296	7.540
5/16	0.312	7.937
21/64	0.328	8.334
11/32	0.343	8.731
23/64	0.359	9.123
3/8	0.375	9.525
25/64	0.390	9.921
13/32	0.406	10.318
27/64	0.421	10.715
7/16	0.437	11.112
29/64	0.453	11.509
15/32	0.468	11.906
31/64	0.484	12.303
1/2	0.500	12.700

INCHES		mm
33/64	0.515	13.096
17/32	0.531	13.493
35/64	0.546	13.890
9/16	0.562	14.287
37/64	0.578	14.684
19/32	0.593	15.081
39/64	0.609	15.478
5/8	0.625	15.875
41/64	0.640	16.271
21/32	0.656	16.668
43/64	0.671	17.065
11/16	0.687	17.462
45/64	0.703	17.859
23/32	0.718	18.256
47/64	0.734	18.653
3/4	0.750	19.050
49/64	0.765	19.446
25/32	0.781	19.843
51/64	0.796	20.240
13/16	0.812	20.637
53/64	0.828	21.034
27/32	0.843	21.431
55/64	0.859	21.828
7/8	0.875	22.225
57/64	0.890	22.621
29/32	0.906	23.018
59/64	0.921	23.415
15/16	0.937	23.812
61/64	0.953	24.209
31/32	0.968	24.606
63/64	0.984	25.003
1	1.000	25.400

Machine Serial Number

When calling in to Park Industries for machine support you may be asked to provide your machine serial number. Find your machines serial number and record it into this section for easy reference.

One of the Serial number tags are located on the inside of the left door in the electrical cabinet.





The other Serial number tag is located by the machines main disconnect switch.

Write the serial numbers of your machines here for easy reference.		Machine Model/Name
Machine Serial Number		
Machine Serial Number		
Machine Serial Number		
Machine Serial Number		
Machine Serial Number		
Machine Serial Number		



TRAINING MANUAL

CUSTOMER CARE | EQUIPMENT

Please Contact Customer Care at 800.785.3391

For faster service, have your MACHINE SERIAL NUMBER ready when calling.
Review the previous page to locate machine serial number.

Park Industries®
Precision. Performance. Peace of Mind.

6301 Saukview Dr.
St. Cloud, MN 56303

800.328.2309
parkindustries.com

FOLLOW US